Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 768-774
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Effects of Inlet Velocity of a Bubble Column Reactor on the Fluid Flow Pattern for Biodiesel Production

Sathis Rajah[1], Nurul Fitriah Nasir[1,*]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.083
Received 20 July 2021; Accepted 30 Nov.2021; Available online 25 December 2021

Abstract: Selecting wrong inlet velocity will decrease the efficiency and increase the production cost of biodiesel production since different velocity will have different flow pattern. Therefore, studying the effects of various inlet velocity on the fluid flow pattern using 3m/s, 4m/s and 5m/s in Bubble Column Reactor for transesterification reaction is essential in determining the best inlet velocity. CFD Simulation using ANSYS Workbench (Fluent) was carried out to determine and analyze the flow pattern after two specific Bubble Column Reactor, one with round obstacle and another with hexagonal obstacle was designed using SOLIDWORKS 2019. Bubble Column Reactor with round obstacle with inlet velocity of 3m/s had the best results in terms of flow pattern. Laminar flow has the best fluid flow compared to turbulent flow compared between these two designs. More types and size of obstacles should be studied in order to get a clear understanding on the effect of obstacles on the fluid flow pattern.

Keywords: Fluid Flow Pattern, Bubble Column Reactor, Laminar Flow, Turbulent Flow, ANSYS Workbench (Fluent)

1. Introduction

Fossil fuels are the main energy source in the world as for now and their overexploitation has resulted in the depletion of the source and this in turn will bring a big impact on the future which depends on it solely. This creates an awareness in looking for an alternative source to replace existing technologies such as petrodiesel which is conventional diesel [1]. Biodiesel is a renewable energy source or a biofuel which is free of sulphur and polycyclic aromatic compounds. Therefore, biodiesel will be very useful because it emits less pollutants, greenhouse gases and its renewability. Biodiesel is normally produced by transesterification process [2]. The process is conducted when three moles of methanol is being reacted with one mole of triglyceride producing a mono-alkyl ester which is called biodiesel and also glycerol as the by-product [3]. Bubble column reactor is specially made to produce fatty acid methyl ester (FAME) as the bubbles of superheated methanol vapor was blown into vegetable oil (palm oil) [4] [5]. There are three type of flow regimes in the bubble column which are homogeneous

**Corresponding author: fitriah@uthm.edu.my*

regime, heterogeneous regime, and also slug flow regime [5]. In biodiesel production, a flow of fluid is very important during the whole process of transesterification because it determines the rate of biodiesel production.

The problem, selecting wrong inlet velocity will decrease the efficiency and increase the production cost of biodiesel production because different velocity will have different fluid flow pattern. The purpose of this research is to study the fluid flow pattern in the Bubble Column Reactor for transesterification reaction using various inlet velocities and also by using different obstacles in the reactor. The study is conducted by CFD Simulation using ANSYS Workbench (Fluent) R19.2 to find out the fluid flow pattern in Bubble Column Reactor. Through this research, the best inlet velocity and obstacles in Bubble Column Reactor for better fluid flow pattern were successfully identified so that a better transesterification process can be achieved.

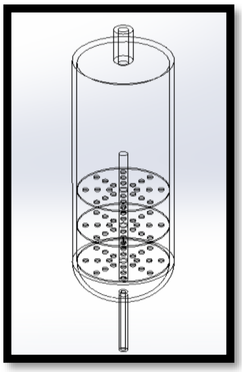
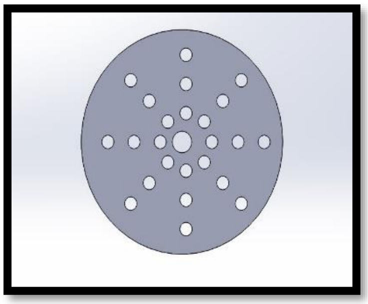
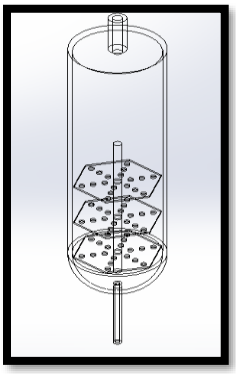
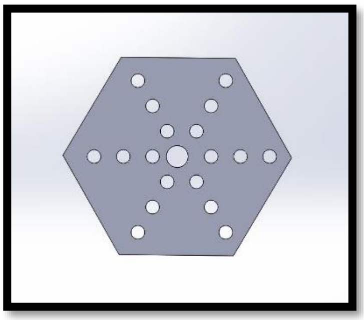
2. Methodology

CFD Simulation method using ANSYS Workbench (Fluent) was used to simulate the Bubble Column Reactor in order to study the flow pattern inside the reactor during transesterification process. Two models of reactor with different obstacles were designed using SOLIDWORKS 2019 software and are tested using three different inlet velocities. The flow pattern obtained were then tabulated and analyzed to come up with a selection of the best design with suitable inlet velocity.

2.1 Geometry Modelling

The two model of the Bubble Column Reactor were designed using SOLIDWORKS 2019 software, one with three round obstacles and another with three hexagonal obstacles. Table 1 shows the design of both Bubble Column Reactor with its obstacles.

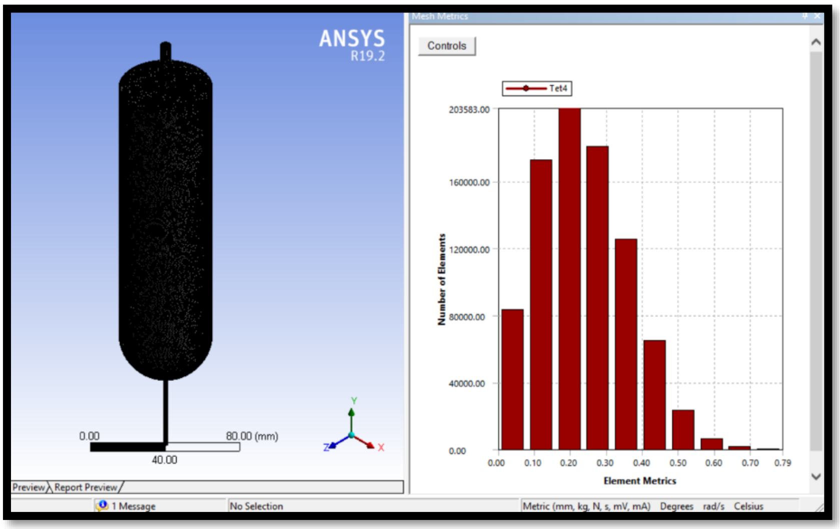
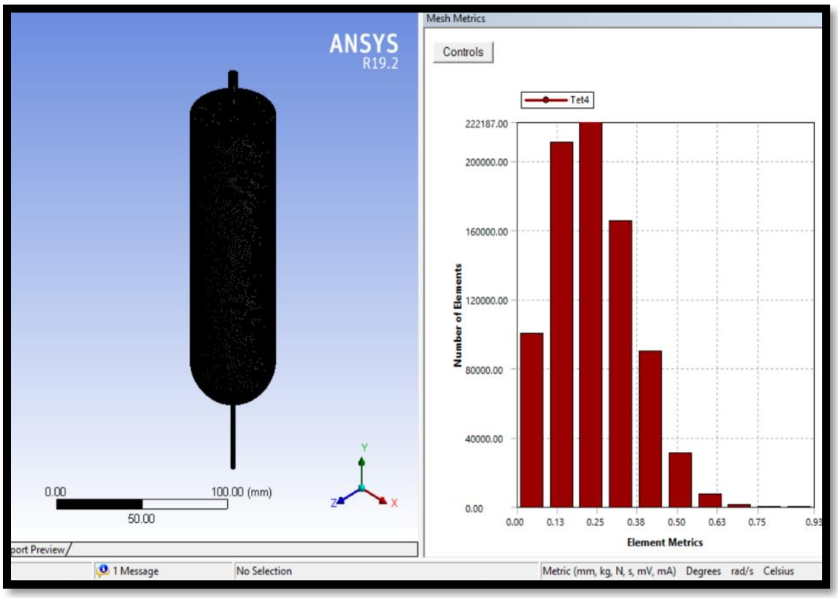
Table 1: Bubble Column Reactor with its obstacles

Bubble Column Reactor	General view	Obstacle view
Model 1 (Round Obstacle)		
Model 2 (Hexagonal Obstacle)		

2.2 Meshing

Table 2 shows the meshed geometry for the designed model and its skewness after meshing was applied to the geometries. Face sizing was done for both model and an overall 5 different parts were conducted face sizing meshing naming inlet, outlet, inner rod, obstacles and the reactor's wall. The mesh metric obtained for model 1 is 0.7891 which is in good range of the skewness mesh metrics spectrum whereas the Orthogonal Quality mesh metric obtained was 0.9932 which is in excellent range of the Orthogonal Quality mesh metrics spectrum. The mesh metric obtained for model 2 is 0.9256 which is in acceptable range of the skewness mesh metrics spectrum whereas the Orthogonal Quality mesh metric obtained was 0.9927 which is in excellent range of the Orthogonal Quality mesh metrics spectrum.

Table 2: Meshed geometry and its skewness

Bubble Column Reactor	Meshing and its skewness
Model 1 (Round Obstacle)	
Model 2 (Hexagonal Obstacle)	

2.3 CFD Simulation

The CFD Simulation in this study was carried out using ANSYS Workbench (Fluent). A combination of Volume of Fluid and k-epsilon model was used so that the flow analysis of multiphase flow can be simulated accurately using appropriate inlet velocities. In order to get a set of equation well-posed, the boundary condition for the Bubble Column Reactor has to be set such as the pressure, temperature and also inlet velocity. A precise boundary condition should be set and defined so that a good and precise result will be obtained. The materials used in this study were air, Methyl-alcohol-vapour, and Palm Oil. The properties of materials and boundary conditions used in the flow analysis were summarized in Table 3 and 4 respectively.

Table 3: Properties of Materials

Properties \ Materials	Air	Methyl-alcohol-vapour	Palm Oil
Density (kg/m3)	1.225	1.43	909
Cp (Specific Heat) (j/kg-k)	1006.43	-	281.0665
Thermal Conductivity (w/m-k)	0.0242	0.0163	0.1041
Viscosity (kg/m-s)	1.7894e-05	1.35e-05	0.069
Molecular weight (kg/kmol)	28.966	32.04	885.4321
Standard State Enthalpy (j/kgmol)	0	-2.01097e+08	-2.056e+09
Reference Temperature (K)	298.15	298.15	298.15

Table 4: Boundary Condition used in Fluid Domain

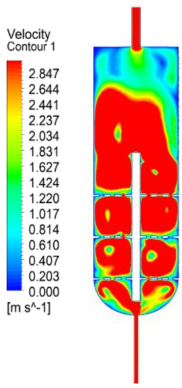
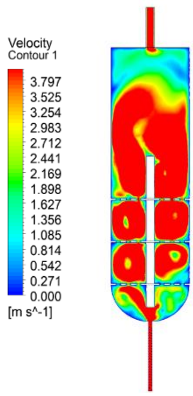
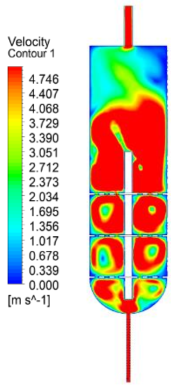
	Inlet Velocity (m/s)		
Parameter	3	4	5
Pressure (Pa)	100000	100000	100000
Temperature (K)	523	523	523
Volume Fraction	1	1	1

3. Results and Discussion

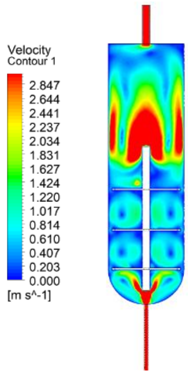
3.1 Contour of Velocity

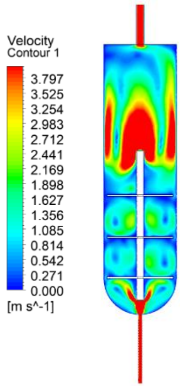
Table 5 below shows the contour of velocity for both model of reactors with the inlet velocities of 3m/s, 4m/s, and 5m/s. As the velocity increases, there is a clear change in the reactor as indicated by the velocity contour and it represents the fluid flow pattern in the Bubble Column Reactor. The dark red colour in the legend shows that maximum velocity is achieved, gradually decreasing velocity until the dark blue colour which is the minimum velocity. The obstacle clearly affects the flow of fluid in the reactor as the 3 obstacles in the reactor disrupting the flow of fluid from the inlet up to the outlet.

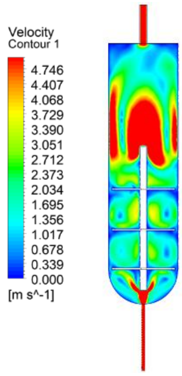
Table 5: Contour of Velocity

	Velocity (m/s)		
Model	3	4	5
1			

2





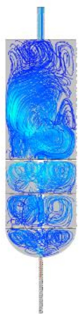
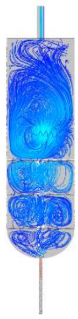
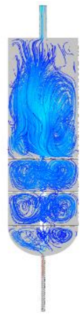


For model 1, the flow in reactor with velocity of 5m/s has been affected the most due to the obstacle because there is a big difference in the contour which indicates there has been a big variation in velocity. It is very clear that the reactor with inlet velocity of 3m/s has more steady flow compared to the reactors with inlet velocity of 4m/s and 5m/s by looking at the contour of velocity and this is indicating that more fluid was allowed to pass through the holes of the round obstacles. For model 2, The reactor with inlet velocity of 3m/s has more steady flow compared to the reactors with inlet velocity of 4m/s and 5m/s by looking at the contour of velocity and this is indicating that more fluid was allowed to pass through the holes and spaces of the hexagonal obstacles.

3.2 Streamline of Fluid Flow

Table 6 shows the Streamline analysis. Streamline analysis is very important in CFD Analysis as it shows the flow of fluid throughout the simulation process and gives us a clear vision on how each and every fluid flow pattern, let it be laminar, transient, turbulent or even circular motion.

Table 6: Streamline of Fluid Flow

	Inlet Velocity (m/s)		
Model	3	4	5
1			
2			

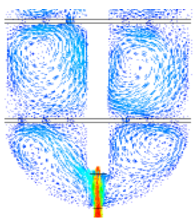
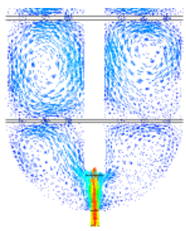
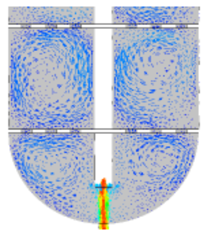
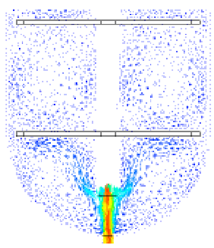
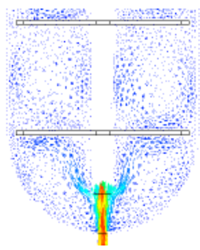
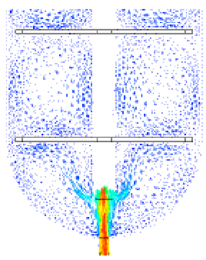
For model 1, it is seen that mixing in reactor with inlet velocity of 3m/s happens more vigorously and happens in more circular motion compared to other velocity. Reactor with inlet velocity of 5m/s does not mix properly and is depicted by the streamline towards the end of the reactor which is more linearly shaped compared to other velocities. Mixing happens in all section of the reactor even though it is separated by three obstacles. The fluid is able to enter and exit through the holes of obstacle smoothly in all section but as the velocity of inlet increases, the lesser the mixing happens in the lowest section of the reactor and this can be justified with the streamline in the lowest section of the reactor becomes lesser.

For reactor 2, it is seen that mixing in reactor with inlet velocity of 3m/s happens more vigorously and in circular motion. It can be justified with the dense of the streamline in the reactor. We can also see that as the velocity increases, the streamline in the lowest section of the reactor decreases. This indicates that the faster the fluid flows into the reactor, the harder for the fluid to mix thoroughly as soon as it leaves the inlet. This is also applying for the highest section of the reactor where the streamlines decrease towards the outlet which is more linear for higher velocities compared to the lower velocities.

3.3 Vector Velocity

Vector analysis is important in CFD Analysis to view the direction of movement of the fluid in the reactor. Table 7 shows the difference in Vector Velocity for both Bubble Column Reactor.

Table 7: Vector Velocity

	Velocity (m/s)		
Model	3	4	5
1			
2			

The results obtained are almost of the same for all velocities in both reactors. The vectors shows that the directional vectors in the centre region is in the reverse direction compared to the wall of reactor. This is because there is a backflow of the fluid until the bottom in the reactor when the mixing process takes up. This indicates that the flow is in turbulent.

4. Conclusion

A combination of fine meshing and high smoothing was required for both reactors to get suitable meshing together with 5 different face sizing meshes for the inlet, outlet, inner rod, three obstacles and the wall of reactor. The lesser the velocity, the more mixing happens in the reactor. The obstacles too had an impact on the flow pattern in the Bubble Column Reactor as there is a clear result showing that a rotational flow happens in the section in between the obstacles which indicates that fluid has entered the sections through the holes and a turbulent flow has been created. As a conclusion, Bubble Column Reactor with Round Obstacle had better result in term of mixing of reactants, velocity throughout the reactor and also directional movement of fluid. To be precise, the reactor with inlet velocity of 5m/s which contains round obstacle had the best outcome of the CFD Simulation Analysis. This also indicates that the design of the reactor will affect the result obtained. For more precise results to be obtained, more variation of inlet velocity should be studied with a bigger deviation of velocities and also more types and size of obstacle should be studied.

Acknowledgement

The authors would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for their support.

References

[1] W. Nurdiyana *et al.*, "Data on greenhouse gases emission of fuels in power plants in Malaysia during the year of," *Data Br.*, vol. 30, p. 105440, 2020, doi: 10.1016/j.dib.2020.105440.

[2] M. Balat and H. Balat, "A critical review of bio-diesel as a vehicular fuel," *Energy Convers. Manag.*, vol. 49, no. 10, pp. 2727–2741, 2008.

[3] B. Najafi, E. Akbarian, S. M. Lashkarpour, M. Aghbashlo, H. S. Ghaziaskar, and M. Tabatabaei, "Modeling of a dual fueled diesel engine operated by a novel fuel containing glycerol triacetate additive and biodiesel using artificial neural network tuned by genetic algorithm to reduce engine emissions," *Energy*, vol. 168, pp. 1128–1137, 2019.

[4] H. Maeda *et al.*, "Biodiesel fuels from palm oil via the non-catalytic transesterification in a bubble column reactor at atmospheric pressure: a kinetic study," *Renew. Energy*, vol. 33, no. 7, pp. 1629–1636, 2008.

[5] N. Kantarci, "F., Borak and KO Ulgen,"," *Bubble Column React. (review)," Process Biochem*, vol. 40, no. 7, pp. 2263–2283, 2005.