Journal of Fundamental and Applied Sciences

Research Article

ISSN 1112-9867

Special Issue

Available online at http://www.jfas.info

ANALYSIS OF TURBULENT FLOW ON DIFFERENT ARRANGEMENT OF BIO-BALLS USING COMPUTATIONAL FLUID DYNAMICS

G. Hayder*, P. Perumulselum

Institute of Energy Infrastructure, Department of Civil Engineering, College of Engineering, Universiti Tenaga Nasional, Selangor, Malaysia

Published online: 24 November 2017

ABSTRACT

This paper aims to study the behavior of an attached growth reactor using CFD simulation. In order to benefit the engineering design of an attached growth reactor, ANSYS Fluent was used to make a simulation and visualize the flow pattern. The main purpose of this simulation is to prevent clogging by understanding the wastewater flow pattern inside the reactor. The investigation has focused on the effects of bio-ball arrangement with different distances in between using computational fluid dynamics and also the effects of bio-ball geometry in water flow pattern. Two different bio-balls were used in the simulation. They are fin ball and spike ball. From the results, it has been proven that arrangement of bio-balls with bigger distance in between have the better distribution of water flow. Better distribution of water flow around the water flow can increase the microorganism growth on the surface of bio-ball.

Keywords: simulation; bio-balls; computational fluid dynamics.

Author Correspondence, e-mail: gasim@uniten.edu.my

doi: http://dx.doi.org/10.4314/jfas.v9i7s.58

1. INTRODUCTION

Attached growth process is getting very famous in research field because of its benefits and advantages in treating the wastewater. Attached growth process is a biological process. Organic matter is removed from wastewater by microorganisms during this process. These microorganisms are primarily aerobic, meaning they must have oxygen to live. Microorganisms attach and grow on the media (materials such as gravel, sand, peat, or specially woven fabric or plastic) that act as the filter in wastewater treatment process. Microorganisms are recycling the dissolved organic material into biofilm that develops on the media [1]. There are favorable circumstances in utilizing the attached growth process. Attached growth process can be maintained easily, consume less energy, and uses less technology. These advantages increase the utilization of attached growth process in small sized communities. This process is very famous in removing the biochemical oxygen demand (BOD), chemical oxygen demand (COD), and suspended particle [2]. Larger area, ineffective during cold season and odor problems are the disadvantages of attached growth processes. Treatment of high organic load wastewater with attached growth process is less effective. The wastewater should undergo preliminary and primary treatment to remove coarse solids and floating debris because the treatment system can be clogged [3]. There are many challenges in attached growth process. The performance of attached growth process will be affected by the types of carrier, carrier filling ratio, hydraulic retention time, sludge retention time, organic loading rate, wastewater characteristics, aeration rate, temperature and etc.

Recently, computational fluid dynamics (CFD) has become widely used for analysis of hydraulic problems in wastewater treatment. It is a tool to investigate the fluid's behavior at every stage in the process and accurately predicting how it will react in an actual casting environment. In doing so, it greatly reduces the time it takes to create a working design, which can then be manufactured in the real world. Because of the work that has been done in the virtual world, the chances of this unit performing as expected are greatly increased and the overall cost of development is substantially reduced. Examples of software that can be used to determine the fluid characteristics in attached growth reactor are ANSYS Fluent, SOLIDWORKS Flow Simulation, AUTODESK CFD and many.

There are many researchers who conducted computational fluid dynamics simulation (CFD) in wastewater treatment process. Rezvani et.al conducted the experimental study and CFD simulation of phenol removal by immobilization of soybean seed coat in an attached growth reactor. They studied the hydrodynamics and reaction behavior in a randomly packed-bed bioreactor. They studied the effect of three different bed porosities under single-phase conditions. The impact of different inlet velocities was also studied. Results from GAMBIT software showed that dimensionless pressure drop for various Reynolds number and removal of phenol from the wastewater at different inlet velocities [4]. Besides that, Le Moullec et.al studied a biological reactor of wastewater treatment with CFD. The CFD study was carried out with FLUENT software. Liquid phase velocity fields and turbulence characteristics have been simulated from the FLUENT software. The results showed that simulated concentration profiles were acceptable for oxygen, nitrate and COD concentration but CFD was unable to simulate ammonium concentration profile [5]. Moreover, CFD simulations were carried out by Yuan Wang et.al in a submerged hollow fiber membrane bioreactor using a porous media approach. They focused the CFD simulations to membrane filtration zone. Inertial loss caused by the hollow fiber bundle was studied with various liquid velocities. The results of the simulation were then applied on the porous media model. They mentioned that the hydrodynamic behavior of the porous media model was improved compared to the hydrodynamic behavior of MBR model [6].

Liquid-solid fluidized beds are commonly used in chemical engineering, food and wastewater treatment industrial applications [7]. CFD simulation was used in fluidized bed reactor (FBR) by Hui Pan et.al. They studied liquid-solid-gas multiphase flow behaviors in FBR. They used CFD simulation for many aspects but significantly hydrodynamics behavior. They also discussed the behavior of bubble in term of formation, shape, and size [8]. Guodong Liu et.al also studied fluidized bed reactor with CFD and Discrete Element Method (DEM). They study the energy dissipation in the gas-liquid fluidized bed reactor. They mentioned that interstitial fluid playing the role in generating energy dissipation. Velocity and inertia of particles are affecting the energy dissipation. Smaller particle velocity has the significant effect of interstitial fluid on the energy dissipation. They also concluded that particles with less inertia can be easily influenced by the effect of interstitial fluid [7].

Membrane filtration is being applied widely in wastewater treatment. The performance of membrane systems decreases because of concentration polarization, fouling and scaling. This causes the membrane system to decrease its productivity by reduced flux and consuming high energy [9]. In a study by Plascencia-Jatomea et.al, CFD simulation has been used for membrane aerated biofilm reactor (MABR). They studied the mass transfer phenomena induced by flow velocity and flow pattern. The detailed flow patterns of the reactor were obtained using COMSOL Multiphysics software. From the results, they concluded that the central region of the MABR corresponds to the channeling zone and the membrane region to the stagnant zone (85% of the MABR volume) can be designed as the region where reaction degradation of pollutants takes place. They mentioned that computational fluid dynamics is a useful tool to determine the ideal flow pattern and reactor design characteristics to produce the ideal flow [10]. Saeed Shirazian et.al also studied the hydrodynamics and mass transfer of membrane reactors. CFD simulation was carried out on hollow-fiber membrane reactors. The simulation was carried out to understand the behavior of membrane reactor in the removal of ammonia. They used UMFPACK software for the simulation and the results showed that ammonia removal decreases in the region near the membrane inlet. Hydrodynamic investigations showed that the velocity reached fully developed after a short distance from the reactor inlet [11].

Furthermore, Shirazi M. M. A. et.al also studied the momentum heat transfer and mass transfer in membrane reactor using CFD. They focused on the optimizing hydrodynamics and effects of various spacers' geometry which they believed one of the causes of reduction in performance of membrane filtration. From the simulation results, they concluded that a fully spacer-filled feed channel for membrane filtration module seems to be the least favorable, due to the reduction in effective membrane area and higher pressure drop [12]. In addition, Mashallah Rezakazemi et.al studied the behavior of membrane contactor using CFD. They stimulated the ammonia removal from industrial wastewater streams using hollow-fiber membrane contactor (HFMC) with COMSOL Multiphysics software. Since the ammonia concentration in the feed tank varies with time, COMSOL was linked with MATLAB to solve both equations of the feed tank and contactor. Results predicted the unsteady state concentration of ammonia in the membrane contactor as well as the feed tank which helps to understand the design of the reactor [13]. Besides hydrodynamic and mass transfer, Recep

Kaya et.al studied the effects of shear stress distribution and pressure loss on hollow fiber membrane modules with CFD. They predicted the relationship between shear stress and membrane module performance. It has been observed that tangential inlet and outlet create rotational flow inside the membrane module which causes higher shear stress which provides better distribution of shear wall stress on membrane surface [9].

In conclusion, CFD is a very productive tool to understand the behavior of a process numerically. CFD can produce many results that are very useful in designing a reactor. By understanding the behavior of a process using CFD simulation, the performance of a process can be maximized. The CFD simulations can be done to understand many aspects such as hydrodynamic behavior, mass transfer, heat transfer, and much more. In addition, multiphase flow behaviors can also be studied in CFD simulation. For instance, the effect of aeration and wastewater flow can be studied together with the effect of suspended solid in CFD simulation. Moreover, the impact of velocity, energy dissipation, shear stress distribution, pressure loss, turbulence characteristics and many other behaviors can be studied too with the help of CFD simulation. Therefore, the CFD simulation is vital to achieving a better performance of a wastewater treatment system. This paper aims to study the behavior of an attached growth reactor using CFD simulation. In order to benefit the engineering design of an attached growth reactor, CFD software, ANSYS Fluent was used to make a simulation and visualize the flow pattern. The main purpose of this simulation is to prevent clogging by understanding the wastewater flow pattern inside the reactor.

2. METHODOLOGY

Two types of bio-balls were used for the simulation study. They are fin ball and spike ball. The two balls were modeled using SOLIDWORKS 2016. An arrangement of 3x3x3 was adopted instead of a single ball to analyze the effect of bulk arrangement on water distribution. Three different distances in between 3x3x3 arrangement of bio-balls were used to analyze the flow pattern around the arrangement. 0.1 mm, 35 mm and 70 mm distances were used to study how the space in between bio-balls affects the flow pattern. Geometry arrangement of a 3x3x3 bio-balls, which was chosen because a plane can be created that cuts the center of the enclosure to analyze the fluid flow around the center of the bio-balls. At the same time, two different bio-balls were used to analyze how the geometry affects the flow pattern. An enclosure with volume of 0.17 m^3 was created in ANSYS Design Modeler. Figure 1 shows the enclosure together with 3x3x3 arrangement of fin ball and spike ball respectively.

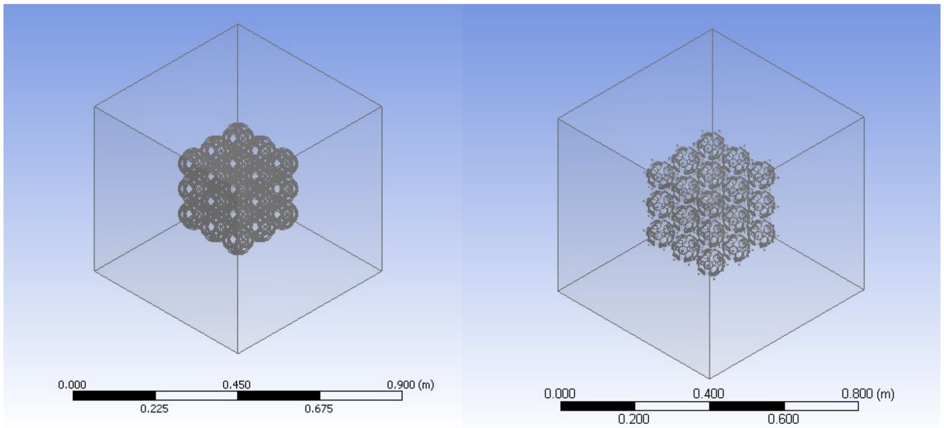

Fig.1. Fin Ball (left) and Spike Ball (right) with Enclosure

Mesh was created with default setting because of limitations in performance of computer. Figure 2 shows the meshing created with ANSYS. Meshing is the process to split the flow domains into smaller subdomains in order to analyse complex fluid flows.

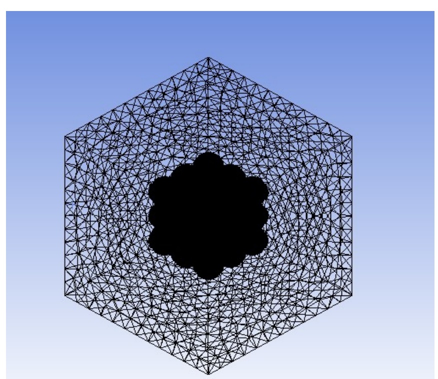

Fig.2. Mesh Created with ANSYS Meshing

The governing equations are then discretized and solved inside of each subdomain. Typically, one of three methods is used to solve the approximate version of the system of equations. The methods are finite element method, finite volume method and finite difference method. The Fluent application uses the finite volume method based solver to compute the results.

The water flow was defined as flowing from right to left of the enclosure. For the single-phase water flow model, the material liquid water was added from the ANSYS Fluent database. The default setting was used for the rest of the setup and as for the boundary conditions, the same water inlet velocity was specified which is 2 x 10^{-6} m/s. The solution was initialized from the water inlet zone and 20 iterations were used to run the simulation. For the fin ball, the calculation was run using 100 iterations while only 20 iterations were used for the spike bio-ball because of the increased computational time that would be needed to try to converge the iterations. The number of iterations is just to get the best-converged solution, for example, the first few iterations would not be smooth but as the number of iterations increases the convergence will be constant with very minimal changes. So, choosing a higher number of iterations will only increase the computational time without adding much value after the first few iterations. Therefore, a number of iteration was set to 20 to obtain an accurate result without compromising on the additional computational time that otherwise would be needed for the convergence of the solution. In the paper by Hadad et al., the iteration has been set up to 1000 because of the shortcomings of the workstation and the absence of ability to deal the high volume of the program [14]. After the simulation completed, water velocity contour was created from CFD post results.

3. RESULTS AND DISCUSSION

The Fluent Post-CFD was able to produce water velocity contour results at the different arrangement of bio-balls. The water velocity contour was analyzed and discussed based on bacterial growth. Figure 3 shows the water velocity contour of two different bio-balls at 0.1 mm distance arrangement.

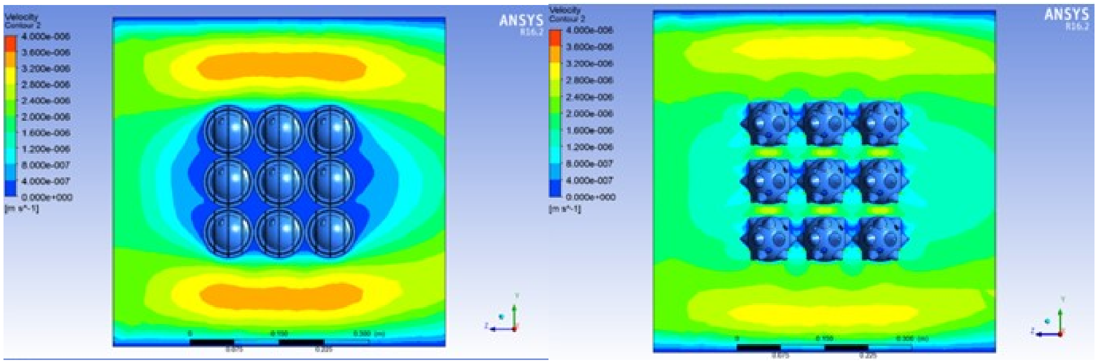

Fig.3. Water Velocity Contours across the 0.1 mm Distance Bio-ball arrangements for fin-ball (left) and spike-ball (right)

The higher velocity of water flow is at the top and bottom of fin and spike ball arrangement, where water inlet being on the left and the water outlet is the opposite. This happened because of the resistance of the bio-balls which causes the water to flow around the geometry of the ball at a higher velocity compared to the initial velocity of the water inlet. Lower velocity of water is at in between the fin and spike ball which was caused due to the effect of resistance from the bulk bio-ball arrangement that resists the water flow in between the bio-balls. As the water is flowing from the left to the right of the enclosure, the water flow was stabilized and velocity has become steady. The dead zones of the bio-balls can be identified by looking at the areas where there are no streamlines passing through as this indicates the water flow is stagnant and the interaction time between the bacteria and wastewater was prolonged that results in a better bacterial growth along this area. Figure 4 and 5 shows the water velocity contour of two different bio-balls at 35 mm and 70 mm distance arrangement respectively.

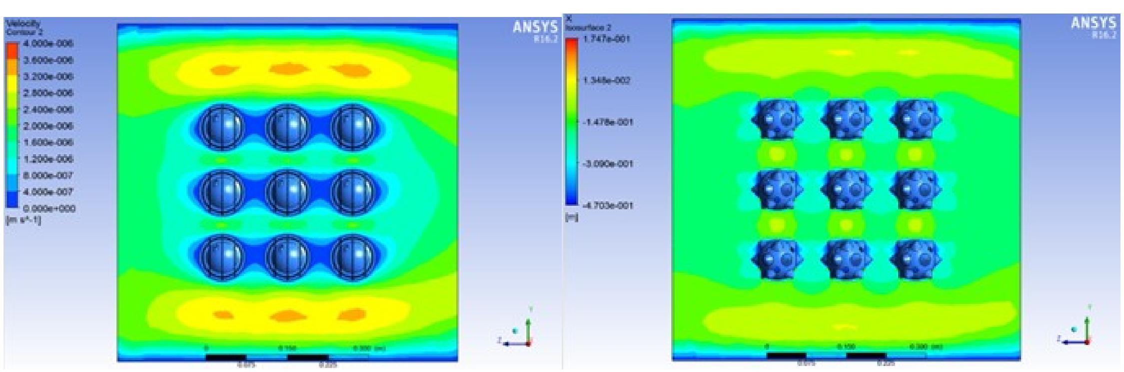

Fig.4. Water Velocity Contours across the 35 mm Distance Bio-ball arrangements for fin-ball (left) and spike-ball (right)

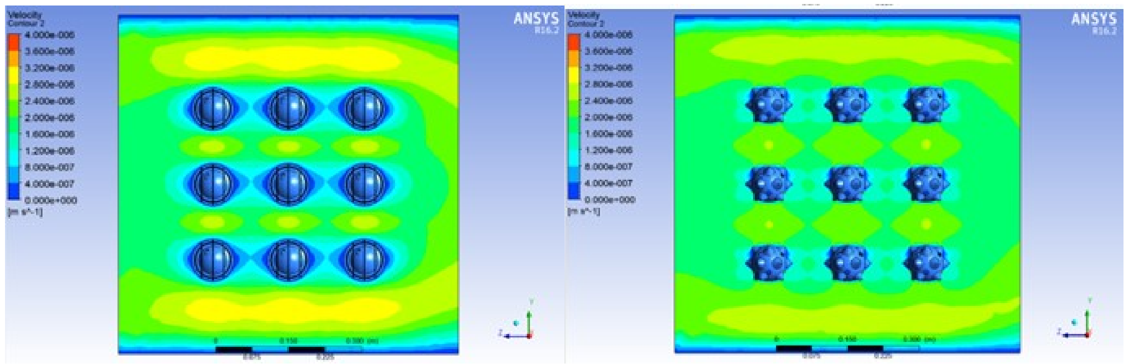

Fig.5. Water Velocity Contours across the 70 mm Distance Bio-ball arrangements for fin-ball (left) and spike-ball (right)

The water flow distribution is almost the same. Since the distance in between the balls increased, the water can flow through the gap and the water distribution is much better compared to 0.1 distance arrangement. This can be due to the decrease in the resistance from the bulk bio-ball arrangement. Water distribution gets better and better when the distance increases.

Graphs of water flow velocity versus Z axis for bio-balls with different distance arrangements were obtained from CFD-Post Results. The results were compared between the two different bio-balls (fin ball and spike ball). Figure 6 shows the velocity versus Z axis for the fin ball, and spike ball respectively with three different distance represented by the colors blue, red and green respectively. It shows the variation in velocity as the water passes through the arrangement of the bio-balls.

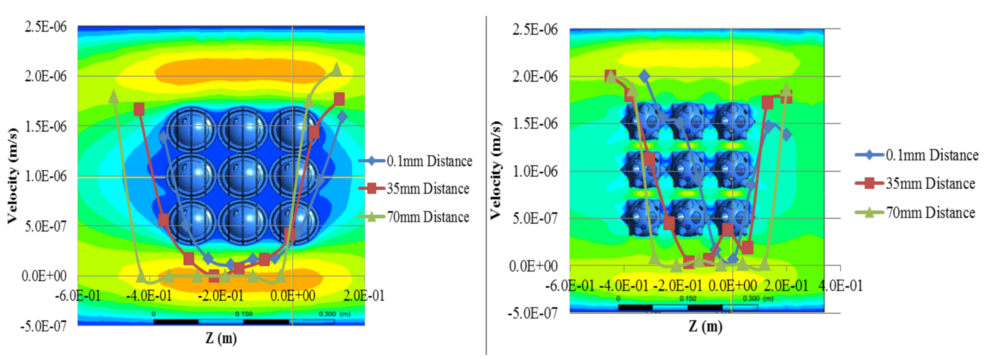

Fig.6. Velocity versus Z axis for Fin-Ball (left) and Spike-Ball (right)

The flow velocity of fin and spike ball with three different distance arrangement along the Z axis generally follow a parabolic curve shape where the water inlet velocity was specified as 0.000002 m/s and it gradually decreases until the water particles flow through fin ball. The water velocity varies from 0.000002 m/s to 0 m/s. For the 70 mm distance arrangement, the first row of the fin and spike balls was located closer to the water inlet followed by the 35 mm distance arrangement and lastly the 0.1 distance arrangement . This explains the water velocity decrease in the 70 mm distance arrangement first, followed by the 35 mm distance arrangement and the 0.1 mm distance arrangement. The water velocity decreases to 0 m/s at some point for 70 mm and 35 mm distance arrangement while it didn't reach 0 m/s for close distance arrangement. This might be due to the arrangement of the bio-balls, where the bulk arrangement causes resistance to the water particles and in turn forces the water particles to flow through a narrow opening and thus increases the water velocity. After the water flow through the bio-balls, the water velocity gradually increases close to the initial water velocity as the resistance provided by the bio-balls was not present anymore. The water velocity changes when it passes through the different arrangement of bio-balls. This creates the non-uniformity of the pressure on the surface of the bio-balls. This can be due to the geometry of the bio-balls. The bio-balls which are positioned at the nearest distance from the water inlet causes a resistance to the water flow and thus the pressure distribution on the surfaces of these bio-balls are higher in contrast to the bio-balls with the furthest distance from the water inlet.

4. CONCLUSION

In the last few years, as a result of increasing availability and accessibility of commercial and open source software suites, the use of CFD has evolved into a robust and precise technique for design, optimization, and control of the wastewater treatment systems. CFD simulations of the water flow characteristics in a wastewater treatment using the standard k-e turbulence model have been carried out using the ANSYS Fluent program. The Fluent program has an ability to simulate the water flow in an enclosure. The single phase water flow velocities are well predicted by the turbulence model in the ANSYS Fluent software. The accuracy of the mean fluid velocity predictions are not strongly influenced by the mesh size used. The changes of velocity in the single phase water flow were presented in the form of graphics such as contours, and line graphs. This validated model will be used to predict and to optimize water flow velocities with aeration of the physical wastewater tank. The investigation has focused on the effects of bio-ball arrangement with different distances in between (0.1mm, 35mm, and 70mm) using computational fluid dynamics and also the effects of bio-ball geometry in water flow pattern. Two different bio-balls were used in the simulation. They are fin ball and spike ball.

From the results, it has been proven that arrangement of bio-balls with bigger distance in between have the better distribution of water flow. Better distribution of water flow around the water flow can increase the microorganism growth on the surface of bio-ball. If the water velocity is only concentrated at a certain point on the surface, it will remove the microorganism at that point. From the CFD simulation of fin and spike ball, it can be clearly seen that water distribution gets better when the distance between the bio-balls increases. The narrow gap between the bio-balls causes the water particles to squeeze through thus increases the velocity. The geometry doesn't affect much on the water distribution. This has been proven from the CFD results of fin and spike ball. The fin ball has slightly better distribution than spike ball since the fin promotes better water circulation. This is due to the fin available at the surface of fin ball. It promotes better circulation of water around it. Besides that, for spike ball, it has spikes on the surface which increases the specific surface area. This will increase the growth of biofilm on it. In conclusion, the bio-balls should be arranged with a bigger distance to increase the water distribution which will increase the bacterial growth as well as attached growth treatment process.

The complete CFD simulation of the flow in the wastewater tank remains a challenge, due to the high CPU and RAM requirements and limited feasibility resulting from the imposed number of iterations needed for the velocity values to converge to the solution. The Fluent program, in the case of having access to computers with a powerful processor, can perform the simulation with a much lesser time for the same amount of iterations. CFD studies in the field of wastewater treatment can, along with their current application as design and troubleshooting tool, be used to develop the next generation of more practical everyday use models.

5. ACKNOWLEDGEMENTS

The author would like to acknowledge the Ministry of Higher Education of Malaysia for the financial support under Fundamental Research Grant Scheme (FRGS).

6. REFERENCES

[1] Hogye, S. The attached growth process an old technology takes new forms. *J. Pipe Line*, 2014, *15*(1), 1-8.

[2] Dabi, N. Comparison of Suspended Growth and Attached Growth Wastewater Treatment Process: A case Study of Wastewater Treatment Plant at MNIT, Jaipur, Rajasthan, India. Europ. J. of Adv. in Eng. and Techn, 2015, 2(2), 102-105.

[3] Azizi, S., Valipour, A., & Sithebe, T. Evaluation of different wastewater treatment processes and development of a modified attached growth bioreactor as a decentralized approach for small communities. The Scientific World Journal, 2013.

[4] Rezvani, F., Azargoshasb, H., Jamialahmadi, O., Hashemi-Najafabadi, S., Mousavi, S. M., & Shojaosadati, S. A. Experimental study and CFD simulation of phenol removal by immobilization of soybean seed coat in a packed-bed bioreactor. *Biochemical Engineering Journal*, 2015, *101*, 32-43.

[5] Le Moullec, Y., Gentric, C., Potier, O., & Leclerc, J. P. Comparison of systemic, compartmental and CFD modelling approaches: application to the simulation of a biological reactor of wastewater treatment. *Chemical engineering science*, 2010, *65*(1), 343-350.

[6] Wang, Y., Brannock, M., Cox, S., & Leslie, G. CFD simulations of membrane filtration zone in a submerged hollow fibre membrane bioreactor using a porous media approach. *Journal of Membrane Science*, 2010, *363*(1), 57-66.

[7] Liu, G., Yu, F., Lu, H., Wang, S., Liao, P., & Hao, Z. CFD-DEM simulation of liquid-solid fluidized bed with dynamic restitution coefficient. *Powder Technology*, 2016, *304*, 186-197.

[8] Pan, H., Chen, X. Z., Liang, X. F., Zhu, L. T., & Luo, Z. H. CFD simulations of gas–liquid–solid flow in fluidized bed reactors—A review. Powder Technology, 2016, 299, 235-258.

[9] Kaya, R., Deveci, G., Turken, T., Sengur, R., Guclu, S., Koseoglu-Imer, D. Y., & Koyuncu, I. Analysis of wall shear stress on the outside-in type hollow fiber membrane modules by

CFD simulation. *Desalination*, 2014, *351*, 109-119.

[10] Plascencia-Jatomea, R., Almazan-Ruiz, F. J., Gomez, J., Rivero, E. P., Monroy, O., & Gonzalez, I. Hydrodynamic study of a novel membrane aerated biofilm reactor (MABR): Tracer experiments and CFD simulation. *Chemical Engineering Science*, 2015, *138*, 324-332.

[11] Shirazian, S., Rezakazemi, M., Marjani, A., & Moradi, S. Hydrodynamics and mass transfer simulation of wastewater treatment in membrane reactors. *Desalination*, 2012, *286*, 290-295.

[12] Shirazi, M. M. A., Kargari, A., Ismail, A. F., & Matsuura, T. Computational Fluid Dynamic (CFD) opportunities applied to the membrane distillation process: State-of-the-art and perspectives. *Desalination*, 2016, *377*, 73-90.

[13] Rezakazemi, M., Shirazian, S., & Ashrafizadeh, S. N. Simulation of ammonia removal from industrial wastewater streams by means of a hollow-fiber membrane contactor. *Desalination*, 2012, *285*, 383-392.

[14] Hadad, H., & Ghaderi, J. Numerical Simulation of the Flow Pattern in the Aeration Tank of Sewage Treatment System by the Activated Sludge Process Using Fluent Program. In *Biological Forum* (Vol. 7, No. 1, p. 382). Research Trend.

How to cite this article:
Hayder G, Perumulselum P. Analysis of turbulent flow on different arrangement of bio-balls using computational fluid dynamics. J. Fundam. Appl. Sci., 2017, *9(7S), 623-635*.