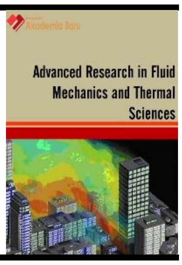
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Numerical Analysis of Velocity Magnitude in a Swirling Fluidized Bed with Spiral Blade Distributor

Open Access

Muhammad Lutfi Abd Latif[1], Mohd Al Hafiz Mohd Nawi[1,*], Wan Azani Mustafa[2], Mohd Sharizan Md Sarip[3], Mohd Aminudin Jamlos[4], Masniezam Ahmad[1], Ku Mohammad Yazid Ku Ibrahim[1], Hazizul Hussein[1]

1 Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis, Aras 1, Blok S2, Kampus UniCITI Alam, Sungai Chuchuh, 02100 Padang Besar, Perlis, Malaysia
2 Department of Electrical Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis, Aras 1, Blok S2, Kampus UniCITI Alam, Sungai Chuchuh, 02100 Padang Besar, Perlis, Malaysia
3 Department of Chemical Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis, Aras 1, Blok S2, Kampus UniCITI Alam, Sungai Chuchuh, 02100 Padang Besar, Perlis, Malaysia
4 Department of Electronic Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis, Aras 1, Blok S2, Kampus UniCITI Alam, Sungai Chuchuh, 02100 Padang Besar, Perlis, Malaysia

ARTICLE INFO

Article history:
Received 2 February 2019
Received in revised form 12 May 2019
Accepted 3 June 2019
Available online 15 July 2019

Keywords:
Swirling fluidized bed; spiral blade distributor; velocity magnitude

ABSTRACT

This paper presents numerical analysis studies to identify air flow distribution affected by various blade distributor configurations in a swirling fluidized bed (SFB). The SFB is in contrast with conventional that impart swirling motion to the particle. The study focused on the effect of spiral blade distributor configuration whereby the effect of various pitch length (60 mm, 80 mm and 100 mm) via various horizontal inclination angle (0°, 12° and 15°). The simulation was applied to compute and evaluate the performance results of velocity magnitude in an SFB. Moreover, the new types of the annular spiral blade had less to reduce the power being wasted during the processes. The numerical analysis with the parameter of horizontal blade inclination of 0° through all pitch length configuration of spiral blade distributor has shown that the air flow in SFB rise until 40 m/s. This condition is due to the air flow was very close to the SFB entry space. The flow of air entering the area between the two blades is not much flow disturbance.

1. Introduction

Fluidization systems are widely used in the chemical industry as it increases solid mixing and also rises the heat and mass transfer. Fluidized bed drying was the most successful approach and normally, the fluidization system is suspended in hot air or stream [1]. Many studies have been made on the fluidized bed to seen the effectiveness which involved with theoretical [2], experiment [3-4] and simulation applications [5-7]. One of the recent application is a gas distributor. Ouyang and

* *Corresponding author.*
E-mail address: alhafiznawi@unimap.edu.my (Mohd Al Hafiz Mohd Nawi)

Levensplel is the first researcher to introduce the spiral distributor [8]. The gas distributor application will induce a uniform and stable fluidization across the bed surface [9]. The main function of the distributor is to distribute uniform fluidizing gas across a layer of the bed. The swirling fluidized bed is one of the fluidization technology that used an annular blade distributor as shown in Figure 1. The blade distributor arrangement is looked like as an array of turbine blade which provides swirling motion inside the bed [10-11]. Compare to the current system, the swirling fluidized beds is fluidizing medium enters the bed at an inclination to the horizontal directed. The swirling fluidized bed is an outcome of studies carried out in order to overcome the disadvantages of the conventional fluidized bed [11-12].

Though a number of bed operating with a similar technique are commercially available such as the rotating and vortexing beds [13], they have not received their due share in terms publication [11]. The main objective of this study is to determine the characteristic of velocity distribution in a swirling fluidized bed affected by various spiral blade distributor configuration in a plenum chamber.

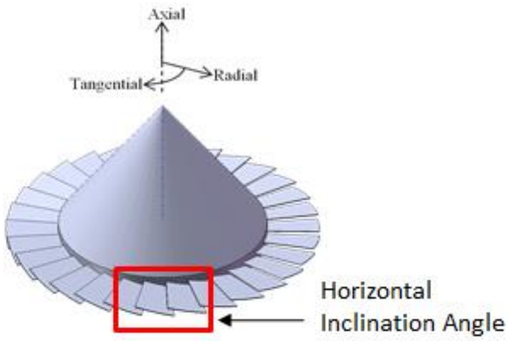


Fig. 1. Annular blade distributor with a conical center body

2. Methodology

2.1 Description of the SFB

Examination of the airflow distribution in an SFB was conducted using commercial CFD software of FLUENT. The computation domain and grid generation were developed via GAMBIT. The two parameters were changed to obtain the correlation between the spiral blade length and the blade horizontal inclination angle as shown in Table 1. The air inlet was modeled as a velocity boundary condition of 19.61 m/s (0.22 kg/s mass flow rate). The horizontal inclination has been selected based on previous studies by [10-11,14].

The full scale model was created due to the previous study. The present study had to change the type of annular blade into a spiral blade distributor which looked like a screw thread. The spiral blade was been designed to cover in all plenum chamber area. The plenum chamber is 600 mm in height while the 300 mm inner diameter. The blades are arrayed in an anti-clockwise direction with 0°, 12° and 15° horizontal inclination and each blade are 1 mm thick. The angle degree of blade distributor has been selected based on the previous study [11,15]. The result from the previous study has shown that angle at 15° leads to high tangential velocity and high uniformly of velocity magnitude. The idea of spiral blade distributor design has been proposed by previous researcher Dr. Mohd Faizal, from Universiti Tun Hussein Onn Malaysia (UTHM). Further, the idea has been realized in these current studies. Thus, to learn more details about the impact of the current design study focusing on pitch length configuration, the value ratio of the diameter of the chamber to the one of the selected blade distributor angle degree of 15° has been used. Starting from 100 mm of pitch length, the value ratio as explains above was been applied to see differentiation. Three pitch length was studied which were 60 mm, 80 mm and 100 mm as shown in Figure 2 below.

Table 1
Spiral blade distributor configuration in a swirling fluidized bed

Case	Cylinder Base Diameter (mm)	Pitch Length (mm)	Horizontal Inclination Angle (°)
1	8	60	0
2		80	
3		100	
4		60	12
5		80	
6		100	
7		60	15
8		80	
9		100	

2.2 Numerical Model

Hence, the same condition setting as previous researchers [14] has been applied in this study. The Tri: Pave Meshing Scheme was applied to the surface and it allowed GAMBIT to create a face mesh consisting of irregular triangular mesh elements. The Tet/Hybrid parameter type that specifies tetrahedral, hexahedral, pyramidal and wedge element were defined to the meshing algorithm. Steady-state segregated implicit solver and Reynolds-Averaged Navier-Stokes (RANS) equation model, RNG k-ε model standard wall treatment were applied to simulate the turbulence flow in the SFB [16]. To reduce numerical diffusion, a second-order upwind scheme was selected for the discretization of the momentum equations [17]. The SIMPLE algorithm was then applied to solve the pressure-velocity coupling algorithms. The mesh elements in the computation domain, as well as the spiral blade distributor, is presented in Figure 3 below.

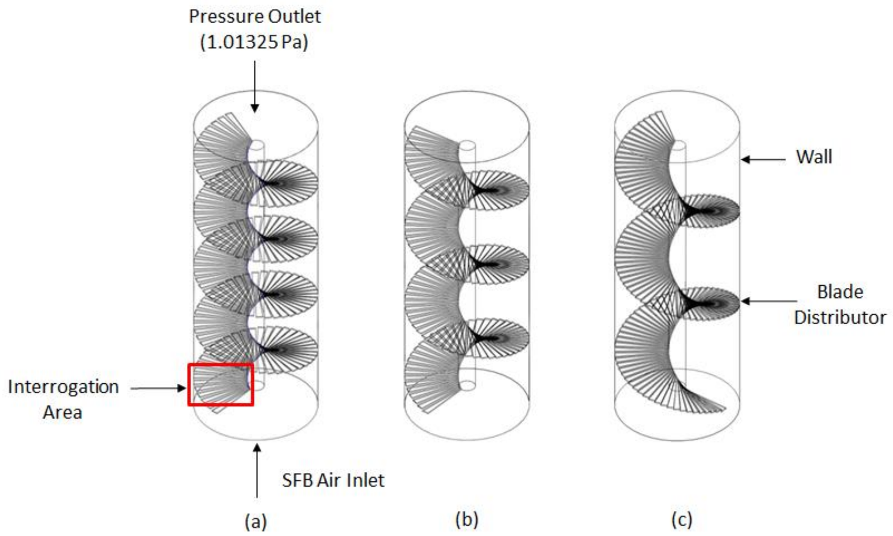


Fig. 2. Configuration of pitch length of spiral blade distributor in a swirling fluidized bed. (a) 60 mm, (b) 80 mm and (c) 100 mm

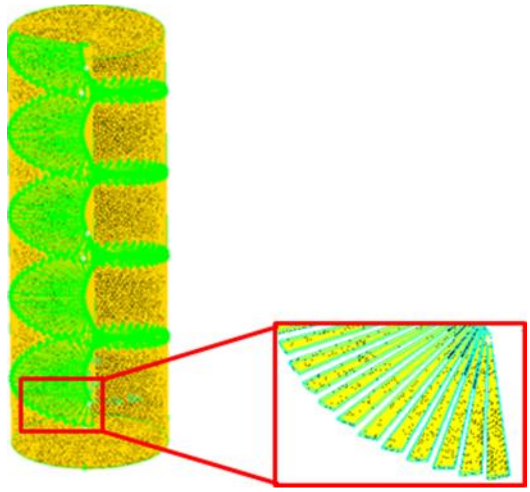


Fig. 3. Computational domain in CFD for spiral blade distributor

The meshing assessment is still the same as the previous study and as the details are as follows [14]. The mesh quality was be evaluated using the Equi Angle Skew (QEAS) criterion, which is lower or equal to 0.2 for more than 95 % of the control volumes. From this evaluation, the mesh quality could be considered satisfactory. In the FLUENT environment, the Reynolds Averaged Navier Stokes (RANS) turbulence equation of the (Re-Normalization Group) RNG methods based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε) which is RNG k-ε model has been selected. This turbulence model is similar to the semi-empirical model namely the standard k-ε model but has an additional term in its dissipation rate (ε) equation that significantly improves the accuracy for rapidly strained flows [17]. Apart from this, it also provides an analytical formula for turbulence Prandtl numbers and also the effect of swirl on turbulence [16].

2.3 Governing Equation

The governing equations [17] for the present study are 3-D momentum and continuity equations in cylindrical coordinates system which were solved for Newtonian, incompressible fluid for in steady flow.

2.3.1 Navier-stoke equation for steady flow case

(r-direction)

$$\rho\left(v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta\partial v_r}{r\partial\theta}-\frac{{v_\theta}^2}{r}+v_z\frac{\partial v_r}{\partial z}\right)=-\frac{\partial P}{\partial r}+\rho g_r+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_r}{\partial r}\right)-\frac{v_r}{r^2}+\frac{\partial^2 v_r}{r^2\partial\theta^2}-\frac{2\partial v_\theta}{r^2\partial\theta}+\frac{\partial^2 v_r}{\partial z^2}\right]$$

(ø-direction)

$$\rho\left(v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta\partial v_\theta}{r\partial\theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right)=-\frac{1}{r}\frac{\partial P}{\partial\theta}+\rho g_\theta+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_\theta}{\partial r}\right)-\frac{v_\theta}{r^2}+\frac{\partial^2 v_\theta}{r^2\partial\theta^2}+\frac{2\partial v_r}{r^2\partial\theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right]$$

(z-direction)

$$\rho\left(v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta \partial v_z}{r\partial\theta}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{\partial^2 v_z}{r^2\partial\theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]$$

2.3.2 Continuity equation

(Cylindrical coordinates)

$$\frac{\partial\rho}{\partial t}+\frac{1}{r}\frac{\partial(\rho r u_r)}{\partial r}+\frac{1}{r}\frac{\partial(\rho u_\theta)}{\partial\theta}+\frac{\partial(\rho u_z)}{\partial_z}=0$$

3. Results

3.1 Resultant Velocity Distribution

This section discusses the results obtained from the numerical analysis study. In this present study, the only the velocity distribution of the system was studied. Data were extracted on a horizontal plane, 32 mm above the SFB inlet and the location was being standardized for each inclination angle according to the pitch length respectively.

3.1.1 The effect of spiral blade distributor on the velocity magnitude

Due to the axial entry plenum chamber with various pitch length configuration, the air flow was deflected by the spiral blade distributor. The resultant velocity profile of various configurations is shown in Figure 4 below. The air flow is rise higher to the bed wall. The same effect of centrifugal force by the previous researcher [11] which at the outer of the cylinder base the air flow causes the swirling and transfer the air into the mass to the wall plenum chamber. This phenomenon will occur in accordance with the rules of physics where the centrifugal force will outward of the original axis [18]. Moreover, the present study has not shown any uniformity occurred even the larger blade angle has been applied on this study as was the case in the study that uses an annular blade distributor [11],[14]. The results of this study agree upon a smaller blade angle will lead to higher jet velocity in the horizontal direction as the previous study [11].

3.1.2 The effect of horizontal inclination angle on the velocity magnitude

Based on the previous study [11,15], the effect on velocity magnitude is proportional to the blade angle, in which higher blade angles result in higher magnitudes. However, in the current study, the result hasn't shown up that phenomenon as shown in Figure 4 at 15° horizontal inclination degree. All velocity magnitude just play around below than velocity of 30 m/s. This is due to the high velocity entry and the trapezoidal area has a wide area for air through it. As Figure 4 above, the effect of 12° degree inclination hasn't shown the same trend as another angle. For example, pitch length at 80 mm the velocity has changed and experienced a fall at blade length of 0.025 m. This is because, when the extreme vertical velocity of turbulence occurs shortly after the blade and this velocity slightly affects the air velocity to become centrifuged. Moreover, when the opening area (trapezoidal) between blades is greater than the centrifugal force becomes weak.

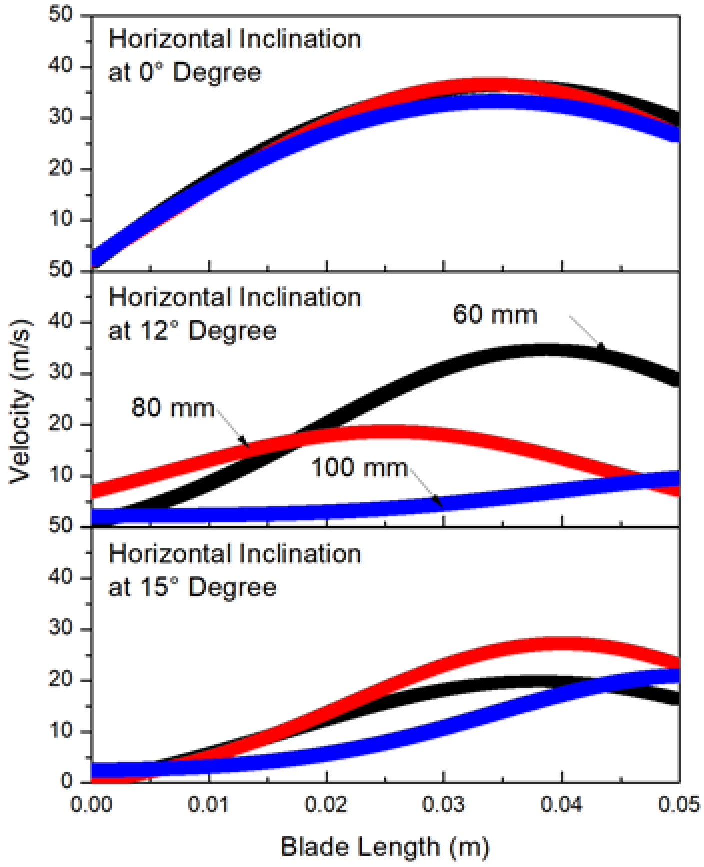


Fig. 4. Velocity magnitude at 32 mm above of SFB inlet with a various horizontal inclination (0°, 12° and 15°) through the pitch length configuration

4. Conclusions

Numerical analysis is an effective method to understand the complex airflow phenomena in SFB especially on velocity magnitude which has covered all three velocity components - axial, tangential and radial. This velocity magnitude reflects the particular design of the spiral blade distributor configuration that excellent mixing can be obtained. From the current arrangement, it was found that distributor with 0° horizontal inclination via all pitch length configuration is the best considering of air flow distribution. This configuration has shown the velocity rise up until 40 m/s. This condition is due to the air flow was very close to the SFB entry space. The flow of air entering the area between the two blades is not much flow disturbance.

Acknowledgment

A sincere thanks also goes to Dr. Mohd Faizal Mohideen Batcha from Universiti Tun Hussein Onn Malaysia (UTHM), Malaysia for his contribution in excellent ideas and guidance on fluidization systems research.

References

[1] Assari, M. R., H. Basirat Tabrizi, and E. Najafpour. "Energy and exergy analysis of fluidized bed dryer based on two-fluid modeling." *International Journal of Thermal Sciences* 64 (2013): 213-219.

[2] Kaewklum, Rachadaporn, and Vladimir I. Kuprianov. "Theoretical and experimental study on hydrodynamic characteristics of fluidization in air–sand conical beds." *Chemical Engineering Science* 63, no. 6 (2008): 1471-1479.

[3] Kaewklum, Rachadaporn, and Vladimir I. Kuprianov. "Experimental studies on a novel swirling fluidized-bed combustor using an annular spiral air distributor." *Fuel* 89, no. 1 (2010): 43-52.

[4] Kuprianov, Vladimir I., Rachadaporn Kaewklum, Kasama Sirisomboon, Porametr Arromdee, and Songpol Chakritthakul. "Combustion and emission characteristics of a swirling fluidized-bed combustor burning moisturized rice husk." *Applied Energy* 87, no. 9 (2010): 2899-2906.

[5] Othman, Safiah, Abas A. Wahab, and Vijay R. Raghavan. "Statistical analysis on the design of flow modifying centre-bodies in a plenum chamber." *CFD Letters* 1, no. 2 (2009): 78-86.

[6] Othman, Safiah, Abas A. A. Wahab, and Vijay R. Raghavan. "Validation by PIV of the numerical study of flow in the plenum chamber of a swirling fluidized bed." *CFD Letters* 2, no. 2 (2010): 85-96.

[7] De Wilde, Juray, and Axel de Broqueville. "Experimental investigation of a rotating fluidized bed in a static geometry." *Powder Technology* 183, no. 3 (2008): 426-435.

[8] Ouyang, Fan, and Octave Levenspiel. "Spiral distributor for fluidized beds." *Industrial & Engineering Chemistry Process Design and Development* 25, no. 2 (1986): 504-507.

[9] Yang, Wen-ching, ed. *Handbook of fluidization and fluid-particle systems*. CRC press, 2003.

[10] Batcha, M. F. M., and V. R. Raghavan. "Experimental studies on a swirling fluidized bed with annular distributor." *Journal of Applied Sciences* 11, no. 11 (2011): 1980-1986.

[11] Faizal, Mohd, Md Seri Suzairin, Mohd Al-Hafiz, and Vijay Raj Raghavan. "CFD studies on velocity distribution of air in a swirling fluidized bed." In *Advanced Materials Research*, vol. 468, pp. 25-29. Trans Tech Publications, 2012.

[12] Sreenivasan, Binod, and Vijay R. Raghavan. "Hydrodynamics of a swirling fluidised bed." *Chemical Engineering and Processing: Process Intensification* 41, no. 2 (2002): 99-106.

[13] Batcha, Mohd Faizal Mohideen, Sulastri Sabudin, and Jamal Hazri Zakaria. "Improvement of Solid-Gas Interaction in Fluidized Bed Systems via Secondary Air-Injection." In *MATEC Web of Conferences*, vol. 135, p. 00014. EDP Sciences, 2017.

[14] Hafiz, M. A., Mohd Faizal Mohideen Batcha, and Norzelawati Asmuin. "Effect of plenum chamber depth in a swirling fluidized bed." In *IOP Conference Series: Materials Science and Engineering*, vol. 50, no. 1, p. 012021. IOP Publishing, 2013.

[15] Batcha, Mohd Faizal Mohideen, M. A. M. Nawi, Shaharin Anwar Sulaiman, and Vijay R. Raghavan. "Numerical investigation of airflow in a swirling fluidized bed." *Asian Journal of Scientific Research* 6, no. 2 (2013): 157-166.

[16] Othman, Safiah, Abas A. Wahab, and Vijay R. Raghavan. "Numerical study of the plenum chamber of a swirling fluidized bed." In *Proceedings of International Conference on Mechanical and Manufacturing Engineering*, pp. 21-23. 2008.

[17] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. *An introduction to computational fluid dynamics: the finite volume method*. Pearson education, 2007.

[18] Cengel, Y.A. and Cimbala, J.M. *Fluid Mechanics: Fundamentals and Applications*. 3rd Ed. McGraw-Hill, 2013.