Effect of twist blade distributor on velocity distribution in a swirling fluidized bed

M A M Nawii, Muhammad Ikman Ishakii, M U Roslii, Siti Nor Suhaidai, Mohd Riduan Jamalludini, Siti Nor Azreen Ahmad Termizi and C Y Khor

1 Simulation and Modelling Research Group (SimMReG), Faculty of Engineering Technology, Universiti Malaysia Perlis (UniMAP), 02100 Padang Besar, Perlis, Malaysia
2 Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis (UniMAP), 02100 Padang Besar, Perlis, Malaysia
3 Department of Chemical Engineering Technology, Faculty of Engineering Technology, Universiti Malaysia Perlis (UniMAP), 02100 Padang Besar, Perlis, Malaysia

Email: alhafiznawi@unimap.edu.my

Abstract. Swirling Fluidized Bed (SFB) is one of the liquid-solid interaction in fluidization improved from conventional systems. This system is usually viewable in the power generation, chemical industry, material production and drying processes. Inside the SFB, there is annular blade distributor which will cause the air to pass through and create a swirl motion on the bed. The energy consumption of a fluidized bed system depends on the distributor's design. The current study has purpose a new design that applies to existing designs of blade inclination angle. The simulation study was conducted using Computational Fluid Dynamics (CFD) to obtain the result of velocity distribution and pressure drops on various blade distributor designs. This study uses two (2) twist angle (80° and 100°) via number of blade distributor (40, 50 and 60). In this study, tangential velocity is the main velocity component by reason of the velocity represents the rotating air velocity in fluidization system. Overall, the design of the 100° twist angle and 40 blades distributor are the best distributors of blades compared to others.

1. Introduction
Fluidization which is known as a physiochemical phenomenon conveys the idea of transforming a fine solid bed into a fluid-like state by passing a gas. Gas flowing through an inlet exerts drag force on the particles and with the increase of gas flow the force exhibited may be large enough that it effects the arrangement of particles inside the bed [1]. Fluidized beds have been known to be used in a variety of applications namely in heat recovery, metal surface treatment, gasification, particle drying, oxidation, endothermic and exothermic, pallet coating, and etc [1].
The Swirling Fluidized Bed (SFB) was developed to improve the performance of conventional fluidized beds. Unlike conventional fluidization, the gas enters the bed at a horizontal angle directed by a suitable distributor [2]. In this case, the annular spiral distributor is used. The purpose of this study is to investigate the gas flow characteristics in the fluidized bed. A great understanding of this study is needed to improve the design of the airflow. Improved air flow design provides low pressure drop where optimum velocity distribution performance can be achieved. The previous study on air flow characteristics focused on distributor using a certain number of blade with different horizontal and radial inclination as well as the aerodynamic characteristics in the plenum chamber. This research is an extension than the previous study in terms of the design. The objectives of this research are: (i) to investigate the aerodynamics characteristics of swirling fluidized bed with various design of distributors and (ii) to investigate the effects of the axial entry of plenum chamber to the air distribution using various blade distributor configurations obtained from the simulation.

1.1. Research Motivation
Swirling Fluidized Bed (SFB) is an innovation from fluidization system. The SFB design have a variant with an inclined of annular bed distributor which is gas injection flow through the blade [3]. Although equipment operating with swirling principles is commercially available, the research involved with distributor and the type of plenum chamber are still scanty. The improvement of SFB process can be done by focusing on various parameter such as the number of blades, blade inclination angle, annular distributor width and bed aspect ratio such as height and diameter of the plenum chamber. One of the main advantages of the SFB is it possesses a low distributor pressure drop. In order to obtain an optimum performance for the SFB, the design of the distributor can affect the pressure drop value. This distributor consists two types of blade inclination angle which are horizontal and radial inclination angle. In the previous study, it was proved that distributor with 15° of horizontal inclination angle was a best design to provide a good performance in fluidization [4].

The new design of distributor is being upgraded in order to produce a distributor that can produce a low pressure drop value and create a tornado effect in a short time. A research has been done by [5] on inclining the distributor in a radially inward direction. The result of this research tells that distributor with 13.5° radial inclination angle give the shortest time for the creation of tornado. But, the pressure drop of this type of distributor is still high. The success of this research can provide fresh and new information to learn more about this technology as SFB is used in many processes in the mineral industry. However, a good distributor design is still the main topic of discussion. In this research, a numerical study using CFD packages was applied caused by the highly complex nature of the flow.

2. Methodology
2.1. Description of the swirling fluidized bed
As reference the previous study [4] by using commercial CFD software of FLUENT the air flow distribution in a SFB has been examine. Further, another commercial CFD package of GAMBIT has
been used to develop the computation domain and grid generation of the SFB designs. In this study, the number of blades distributor 40, 50 and 60 as well as various twist angle are used to simulate the plenum chamber design as shown in Figure 3. Each case of the angle blade also varied and it will be explained in the next topic. There are three type of angles that have been used in this study, radial inclination, horizontal inclination and twist angle. A constant angle of horizontal inclination 15° and radial inclination 10° were been set in this present study. A variation angle of 80° and 100° are being used for twist angle that applied in each case of blade distributor. The two parameters were changed to obtain the correlation between the twist angle of blade distributor and the number of blade distributor as shown in Table 1. The air inlet was modelled as velocity boundary condition of 2.25 m/s. The horizontal inclination has been selected based on previous studies by [6] - [8]. The full scale model was created due to the previous study. The present study had change the type of blade into twist angle of blade distributor which looked like a turbine blade. The blade has been design and placed at one level in plenum chamber area only. The plenum chamber is 600 mm in height while the 300 mm inner diameter [4]. The blades are arrayed in clockwise direction with 15° [8] horizontal inclination and radial inclination 10° [5] and each blade is 1 mm thick. The angle degree of blade distributor has been selected based on the previous study [7][9]. The result from previous study has shown that angle at 15° leads to high tangential velocity and high uniformly of velocity magnitude. Thus, to learn more details about the impact of the current design study focusing on twist angle of blade configuration, the value ratio of the diameter of the chamber to the radius blade distributor (50 mm) was been used.

2.2. Numerical model
Hence, same condition setting as previous researchers [4] and [8] 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 $k$-$\varepsilon$ model standard wall treatment were applied to simulate the turbulence flow in the SFB [10]. To reduce numerical diffusion, a second-order upwind scheme was selected for the discretization of the momentum equations [11]. The SIMPLE algorithm was then applied to solve the pressure-velocity coupling algorithms. The mesh elements in the computation domain as well as the twist angle of blade distributor is presented in Figure 2 below.

| Case | Twist Angle of Blade Distributor (°) | Number of Blades Distributor (n) | Horizontal Inclination (°) | Radial Inclination (°) |
|------|-------------------------------------|----------------------------------|---------------------------|-----------------------|
| 1    | 80                                  | 40                               | 15                        | 10                    |
| 2    | 100                                 |                                  |                           |                       |
| 3    | 80                                  | 50                               | 15                        | 10                    |
| 4    | 100                                 |                                  |                           |                       |
| 5    | 80                                  | 60                               | 15                        | 10                    |
| 6    | 100                                 |                                  |                           |                       |
Figure 2. Volume meshing and example of grid in the Swirling Fluidized Bed; (a) Blade distributor and (b) Twist angle

The meshing assessment is still the same as the previous study and as the details are as follows [4]. The mesh quality was be evaluated using the Equi Angle Skew (QEAS) criterion, which are lower or equal to 0.2 for more than 95 % of the control volumes [4]. From this evaluation the mesh quality could be considered satisfactory. In 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 [4][11]. This turbulence model is similar to the semi-empirical model namely standard k-ε model but has additional term in its dissipation rate (ε) equation that significantly improves the accuracy for rapidly strained flows [11]. Apart from this, it also provides an analytical formula for turbulence Prandtl numbers and also the effect of swirl on turbulence [10].

2.3. Governing equation

The governing equations [11] 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

(r-direction)

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

(θ-direction)

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

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

2.3.2. Continuity Equation

(z-direction)

\[ \frac{\partial \rho}{\partial t} + \frac{1}{r} \frac{\partial (\rho u_r)}{\partial r} + \frac{1}{r} \frac{\partial (\rho u_\theta)}{\partial \theta} + \frac{\partial (\rho u_z)}{\partial z} = 0 \]

3. Results and discussion

3.1 Resultant velocity distribution

This section discusses the results obtained from the numerical analysis study. In this present study, only velocity distribution (velocity magnitude) of the system were studied. Data was extracted on a horizontal plane, 20 mm above the SFB inlet and the location was being standardized for each twist angle of blade distributor.

3.1.1 The effect of twist blade distributor on velocity magnitude

The distributor blade deviates the air as the air enters the distributor. It can trigger the impact of swirling in the chamber of the plenum. Due to centrifugal force, the swirling impact may cause the air to mass at the exterior edge of the column. Thus, Figure 1 in section 1 shows all of the three (3) velocities components of the air flow obtained. The air flow acquired are split into (a) tangential velocity, (b) radial velocity, and (c) axial velocity. In real industrial application, the fluidization is created by axial velocity while swirling effect is created by tangential velocity. Swirling gas created the centrifugal force and lead to existence of radial velocity. However, this current study only presents the result of velocity magnitude only.

The annular blade type represents the different location on the plane 20 mm above the distributor at 80° and 100° twist angle and it is essential at particular periphery to differentiate the distribution of velocity. Additionally, distributors with a large amount of blades contribute to the uniform velocity of the system like as shown in Figure 3 to Figure 5. The graph in Figure 3 to Figure 5 shows the velocity distribution of the velocity magnitude for each case and the result shows as the twist angle decreases, the resulting velocity of high air flows is uniformity. When the twist angle decreases, the resulting velocity tends to diverge more and in this case, the number of 50 blades distributor with 80 ° twist angle has a lower velocity deviation among other instances. Although the number of 60 blade distributors to spread the velocity magnitude in great value as compared to others as shown in Figure 5. Uniformity in velocity magnitude is an important thing in swirling fluidized bed because the particles or solids in the bed may have been affected on dried. Initially, the motion of the bed particles is completely mixed simultaneously and other particles are partly processed at the same time. As from previous study the high tangential velocity in the fluidized bed leads to high swirling motion causes a high effect of tornado. So the result of velocity magnitude was present a part on high value of tangential velocity result compare to the axial velocity and radial velocity values.
Figure 3. Velocity magnitude at different twist angle; (a) 80° and (b) 100° via 40 blades distributor

Figure 4. Velocity magnitude at different twist angle; (a) 80° and (b) 100° via 50 blades distributor
4. Conclusion
As conclusion, the current study have shows that the air velocity distribution will be affected due to various parameter of different twist angle of blade distributor and different number of blade distributor used. High air velocity magnitude will produce high tangential velocity of air. High tangential velocity will produce a better SFB system as it will produced the tornado effect that can help in increasing the rate of processes such as drying. Distributor with higher blade numbers give the highest tangential velocity. Another characteristic of a good SFB system is having low pressure drop. In this selected parameter has shown a very significant parameter on low pressure drop caused by higher fraction of open area (FOA) that allow the air flow to passed through distributor blade easily. The discussion on discharge coefficient or pressure drop will present latter in the other manuscripts.

5. References
[1] Paulose M M *Hydrodynamic Study of Swirling Fluidized Bed and The Role of Distributor*
[2] Wellwood G A 1997 *14th Int. Conf. Fluid. Bed Combustion* 11 618-628
[3] Sreenivasan B and Raghavan V R 2002 *Chem Eng Process* 1 41(2) 99-106
[4] Nawi M A *Aerodynamics of a Swirling Fluidized Bed* (Master Thesis: Universiti Tun Hussein Onn Malaysia)
[5] Batcha M F, Nawi M A, Sulaiman S A, and Raghavan V R 2013 *Asian J. Sci. Res.* 1 6(2) 157-166
[6] Batcha M F M and Raghavan V R 2011 *J Appl Sci.* 11 1980-1986
[7] Faizal M, Seri S M, Al-Hafiz M and Raghavan V R 2012 *Adv. Mat. Res.* 468-471 25-29
[8] Hafiz M A, Batcha M F and Asmuin N 2013 *IOP Conf Ser Mater Sci Eng* 50 1 012021
[9] Safiah O, Wahab A A and V R Raghavan 2010 *CFD Letters* 2(2) 85-96
[10] Versteeg H K and Malalasekera W 2007 *An Introduction to Computational Fluid Dynamics* (Essex: Pearson)
[11] Nawi M A, Amin M R, Kasim M S, Izamshah R, Ishak M I, Khor C Y, Rosli M U, Jamalludin M R, and Syafiq A M 2019 *IOP Conf Ser Mater Sci Eng* 551 1 012106
[12] Latif M L, Nawi M A, Mustafa W A, Md M S, Sarip M A, Ahmad M, Ibrahim K M, and Hussein H 2019 *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 59 1 38-44
Acknowledgements
The authors acknowledge the facilities and the staff involvement, Wan Mohd Akmal Wan Makhdzar from the Kolej Vokasional Kangar, Kangar, Perlis.