Journal of Advanced Research in Applied Sciences and Engineering Technology 29, Issue 2 (2023) 1-11

Journal of Advanced Research in Applied Sciences and Engineering Technology

Journal homepage:
https://semarakilmu.com.my/journals/index.php/applied_sciences_eng_tech/index
ISSN: 2462-1943

Twist Blade Distributor in Fluidization Systems: Part 1 – The Computational Procedure

Mohammad Azrul Rizal Alias[1], Mohd Al Hafiz Mohd Nawi[2,4,*], Mohd Sharizan Md Sarip[3], Md Tasyrif Abdul Rahman[1], Muhamad Silmie Mohamad Shabri[1], Raja Muhammad Zulkifli Raja Ibrahim[1]

[1] Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), Kampus Alam UniMAP, Pauh Putra, 02600 Arau, Perlis, Malaysia
[2] Simulation and Modelling Research Group (SiMMREG), Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), Kampus Alam UniMAP, Pauh Putra, 02600 Arau, Perlis, Malaysia
[3] Faculty of Chemical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), Kompleks Pusat Pengajian Jejawi 3, Kawasan Perindustrian Jejawi, 02600 Arau, Perlis, Malaysia
[4] Sports Engineering Research Center (SERC), Universiti Malaysia Perlis (UniMAP), Kampus Alam UniMAP, Pauh Putra, 02600 Arau, Perlis, Malaysia

ARTICLE INFO

Article history:
Received 8 October 2022
Received in revised form 6 December 2022
Accepted 28 December 2022
Available online 15 January 2023

Keywords:
Fluidized bed; Twist blade distributor; Velocity magnitude; Tangential velocity

ABSTRACT

Flowing gas in fluidized bed through selected inlet distributor may imparts a drag effect on the particles, would cause an increase in gas flow, that maybe sufficient to rearrange the particles movement. Thus, study on the airflow in a fluidization system through numerical analysis has been conducted to investigate the airflow distribution affected by new model distributor of twist blade distributor configuration. The present study would emphasis on computational procedure and parametric study via ANSYS Fluent before a detailed study on selected twist blade distributor are conducted. The selected parametric study on the twist blade distributor configuration whereby the twist blade angle (100°), horizontal inclination angle (15°), radial inclination angle (10°) and number of blades (60) was carried out. Therefore, the results of the studies that have been carried out meet the expected standards based on previous studies.

1. Introduction

Fluidization systems have widely been used in various industrial sectors such as power generation, chemical industry, material production, drying processes, chemical processing, etc. One of the famous designs among researchers in fluidization systems is a Swirling Fluidized Bed (SFB) [1]. The current of SFB systems would tend to generate swirling motion inside the bed. By comparing to the conventional fluidization systems, present SFB systems possess an annular blade distributor (horizontal and radial inclination) as shown in Figure 1 [2]. According to the previous researchers involved in fluidization systems study has found that a higher number of blade distributors through high blade distributor horizontal inclination would results in higher swirling velocity (tangential

* *Corresponding author.*
E-mail address: alhafiznawi@unimap.edu.my

https://doi.org/10.37934/araset.29.2.111

velocity) and provide a uniform velocity on the bed [2,3]. The fluidization systems create produces a sufficient in swirling motion within the bed, that encourages lateral mixing and very important in a fluidized bed [3]. Additionally, it can fluidize Geldart Type-D particles efficiently. Moreover, it has the capacity to fluidize Geldart Type-D particles, which are often large and difficult to fluidize [4]. As a result of these advantages, SFB has become the most effective technology for several applications, including as heat transfer and bed dynamics, drying process, new modern application and many other [5-11].

Figure 1 shows the annular twist blade distributor used in the current investigation, which is composed of a number of blades that are inclined horizontally and radially. For the blade inclination, the horizontal blade inclination produces swirling-fluidization while the radial blade inclination provides inward momentum, hence enhancing the fluidization systems hydrodynamics. However, to the author knowledge, none of these studies tried to explore the relation between twist blade configuration and the air flow distribution. Prior to this, the examination of the influence on the opening of the blade gap relative to the previous SFB's concentration of blades was conducted.

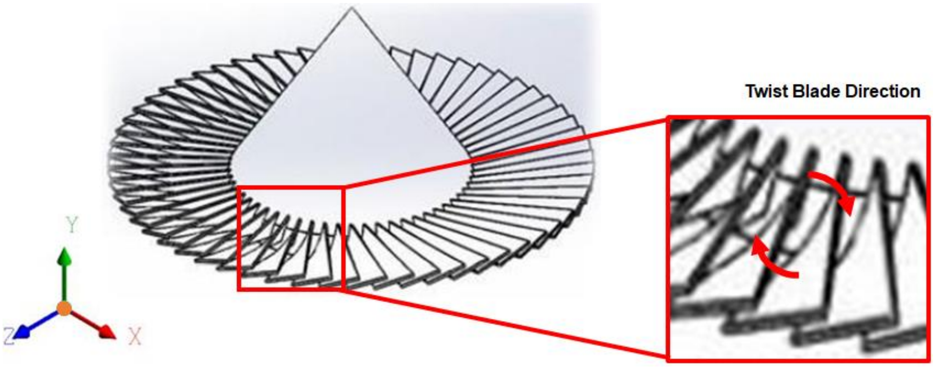


Fig. 1. Annular twist blade distributor with a conical center body

2. Methodology

2.1 Numerical Simulation Process

A flow chart of the research procedures is illustrated in Figure 2. The final step in this study is referred to as the post-processing step using several of graphical methods such as grid, contour, vector, and line plots which will present in next discussion in Part 2.

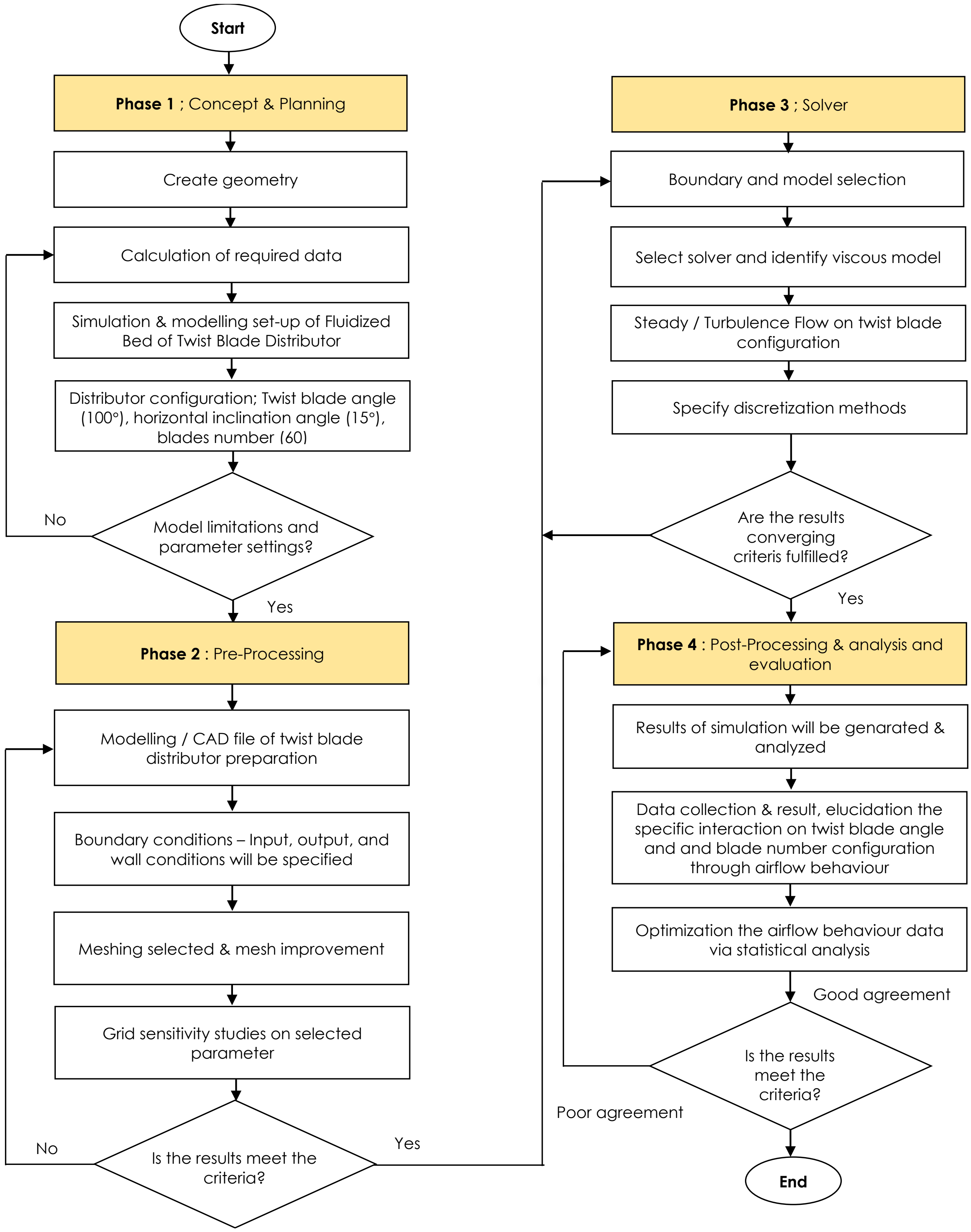


Fig. 2. Flow chart of the research methodology

2.2 Description of the SFB

The airflow distribution, computation domain and grid generation in current fluidization systems model was investigated using ANSYS Fluent commercial Computational Fluid Dynamics (CFD) software. Figure 3 shows how the number of blades distributor 60 is employed in this study. The air inlet was modelled with a 2.25 m/s (0.22 kg/s mass flow rate) velocity boundary condition and the pressure outlet was set at atmospheric pressure (101,325 Pa). As a result, the flow in the present investigation is both constant and incompressible. Thus, as the air flow passes through the geometric volume, the fluid element's density and duration remain constant. In addition, the no-slip shear condition presupposes that the fluid velocity relative to the geometric boundary is zero. In addition, the wall motion was configured to be stationary.

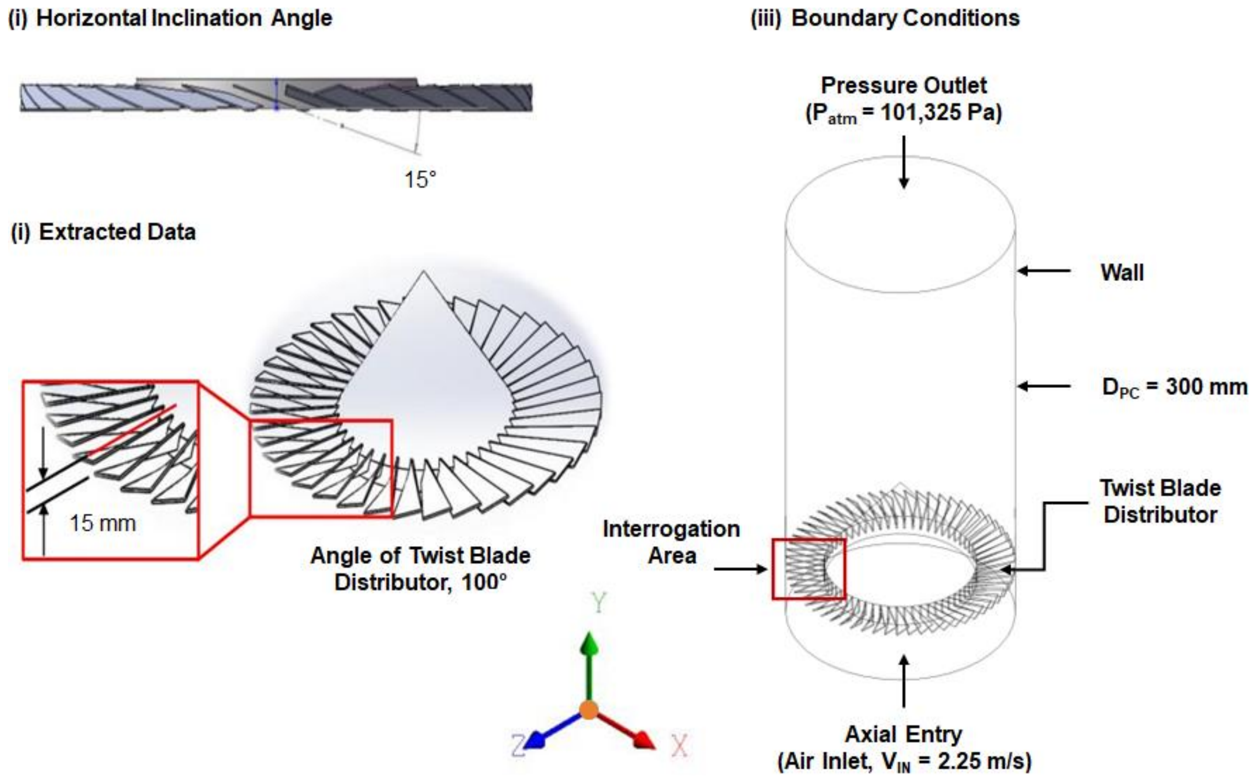


Fig. 3. Fluidization systems via twist blade distributor configuration

Three types of angles were used: radial inclination, horizontal inclination, and twist angle. In this study, a constant horizontal inclination angle of 15°, radial inclination angle of 10°, and twist blade angle of 100° were used. The horizontal inclination and radial inclination were chosen based on previous studies [1-3]. As a result of the prior research, a full-scale model was developed where the blade model was changed to a twist-angle blade distributor that resembled a turbine blade. The blade has been positioned in the plenum chamber area. The plenum chamber has a diameter of 300 mm [1-4]. Each blade is 2 mm thick and is arranged clockwise with horizontal inclination of 15° and radial inclination of 10° [2,3]. Based on past researches by Batcha and Raghavan [2] and Batcha *et al.,* [3], the angle degree of the blade distributor, 15° angle results in a high tangential velocity and a high uniformity in velocity magnitude. To learn more particular about the impact of the present design research centered on the twist angle of blade arrangement, the value ratio of the chamber diameter to the radius blade distributor (50 mm) was chosen. The reference of the data extraction was located 15 mm above the twist blade distributor.

2.3 Numerical Model

As a result, to the current study, the similar condition setting as prior researchers was used [3,12]. ANSYS software was able to construct a face mesh comprised of irregular triangular mesh elements using the Tri: Pave Meshing Scheme, which was applied to the surface and the geometry volume [13].

The meshing algorithm was given the Tet/Hybrid parameter type, which defines tetrahedral, hexahedral, pyramidal, and wedge elements. The turbulent flow in the fluidization systems was simulated using a steady-state segregated implicit solver. 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), also known as the RNG k-ε model, has been chosen in the ANSYS Fluent environment [14].

For the discretization of the momentum equations, a second-order upwind approach was chosen to reduce numerical diffusion [14]. The pressure-velocity coupling algorithms were then solved using the SIMPLE algorithm. The meshing evaluation is the same as in the prior study, with the following details [3]. The mesh quality might be rated satisfactory based on this assessment. This turbulence model is comparable as be shown in the next sub-topic.

2.4 Governing Equation

The current study's governing equations are three-dimensional momentum and continuity equations in cylindrical coordinates that were solved for Newtonian, incompressible fluid in steady flow [14].

2.4.1 Continuity equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

where u, v, and w are velocities in the x, y, and z axes, respectively.

2.4.2 Momentum equation

(x direction)

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial u}{\partial t}+\left[\frac{\partial}{\partial x}\left(\eta\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\eta\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\eta\frac{\partial u}{\partial z}\right)\right]+\rho g_x \tag{2}$$

(y direction)

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho}\frac{\partial P}{\partial y}+\left[\frac{\partial}{\partial x}\left(\eta\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\eta\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\eta\frac{\partial v}{\partial z}\right)\right]+\rho g_y \tag{3}$$

(z direction)

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial P}{\partial w}+\left[\frac{\partial}{\partial x}\left(\eta\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\eta\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\eta\frac{\partial w}{\partial z}\right)\right]+\rho g_z \tag{4}$$

where ρ is density, t is time, P is pressure, η is viscosity, and g_x, g_y, g_z, are gravities in the x, y, and z axes, respectively.

3. Results

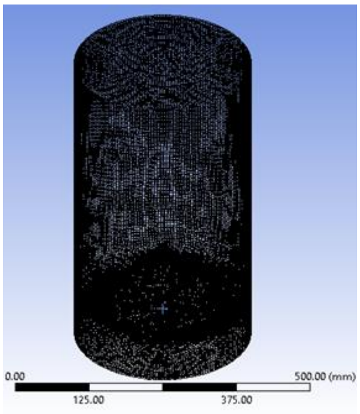
The findings of the numerical analysis investigation are discussed in this section. In this study, the system's velocity magnitude was examined. Since this was in the optimal location to examine the airflow characteristics, information was taken 15 mm above the distributor on a horizontal plane.

3.1 Grid Independence Study

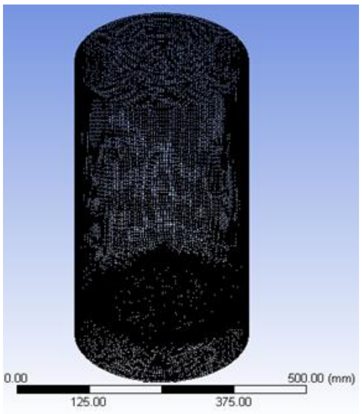
A grid sensitivity study was carried out in order to find an appropriate meshing method that produces minimal computational errors due to truncation and round-off error while taking less time to compute. Logic dictates that a finer mesh will have a smaller truncation error but a greater round-off error, resulting in a longer calculation time [7]. In ANSYS Fluent, five distinct meshing schemes were created, labelled A, B, C, D, and E, each with a different number of cell elements. One of the most crucial performance requirements in fluidization is distributor pressure drop, and this grid's independence from changes in distributor pressure drop is being examined [15]. Table 1 and Figure 4 both offer an overview of the current study. As shown in Figure 4, the sensitivity study conducted shows that the most suitable scheme to be applied to all the case studies is scheme E (which contains cell elements of 530,244). Additionally, the maximum face size, minimum size, and maximum tetrahedral size have all been specified in the current manuscripts' relevant center. Figure 4 illustrates how every strategy had the finest size function.

Table 1
All different scheme meshing parameter

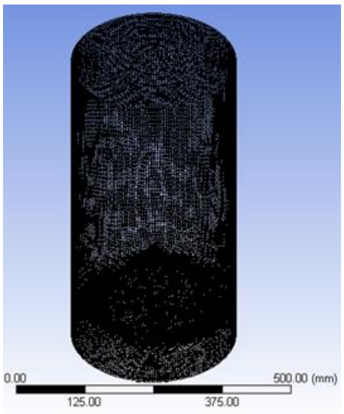
Meshed Region	Relevance Centre	Min Size	Max Face Size	Max Tetrahedral Size
Scheme A	Coarse (Curvature)	0.3 mm	5.0 mm	20.0 mm
Scheme B	Medium (Curvature)			
Scheme C	Fine (Proximity)			
Scheme D	Fine (Fixed)			
Scheme E	Fine (Curvature)			



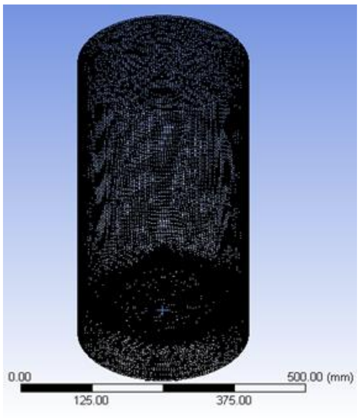
Scheme A
Number of cells: 520,486



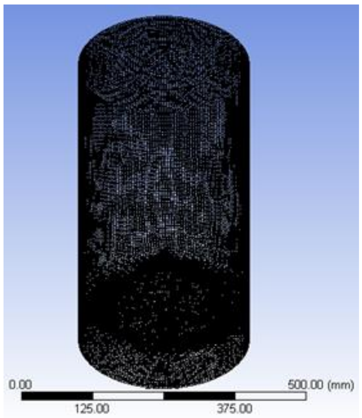
Scheme B
Number of cells: 524,009



Scheme C
Number of cells: 522,122

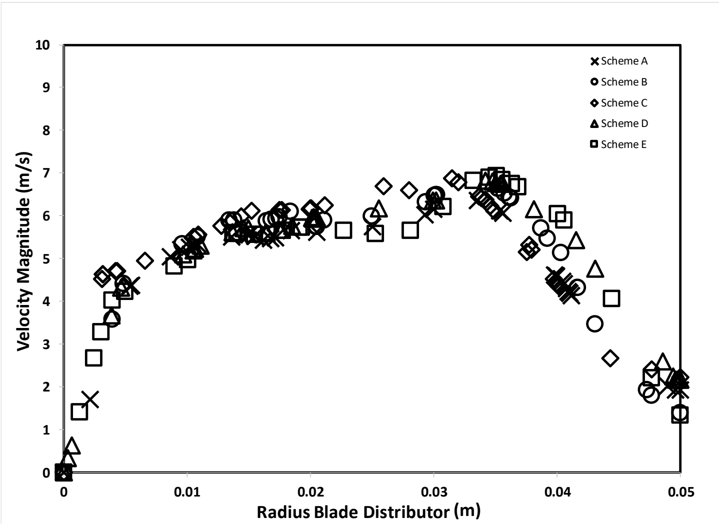


Scheme D
Number of cells: 519,288

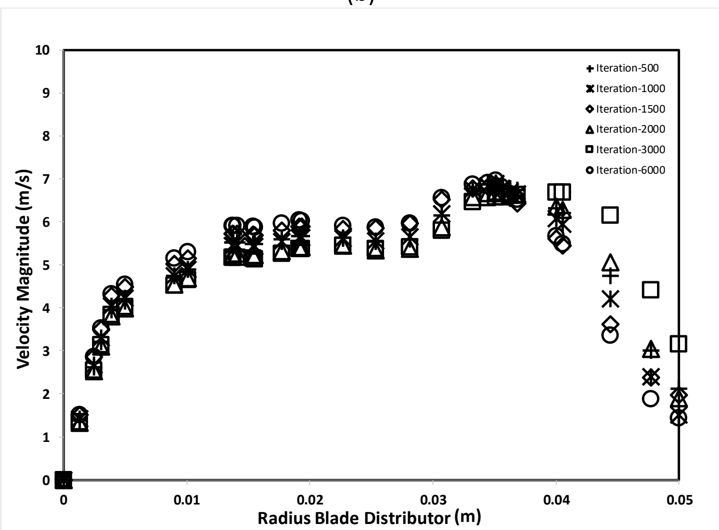


Scheme E
Number of cells: 530,244

(a)



(b)



(c)

Fig. 4. (a) Meshing of fluidization system via twist blade distributor with their respective schemes, (b) Velocity distribution for a different number of grids, (c) Velocity distribution at various number of iterations

3.2 Turbulence Model Selection

All the chosen models (six total turbulent models; five eddy viscosity models and one Reynolds Stress Model (RSM)) were compared in the selection of turbulent models. The eddy viscosity models include the standard $k{-}\varepsilon$, realizable $k{-}\varepsilon$, and RNG $k{-}\varepsilon$ model. Figure 5 has shown that all models produce the best agreement with the current and previous finding, which the velocity tend to deteriorate near to the plenum wall. In fact, the flow in this region exhibits forced vortex behaviour as a result of the vortex's centrifugal effect, which was generated by the blade distributor's inclination angle. This situation will be discussed in more detail (Part 2 – The Air Flow Characteristics). On the other hand, as indicated by Tawfik *et al.,* [7], the RNG $k{-}\varepsilon$ model offers better agreement in the region of the annular blade distributor.

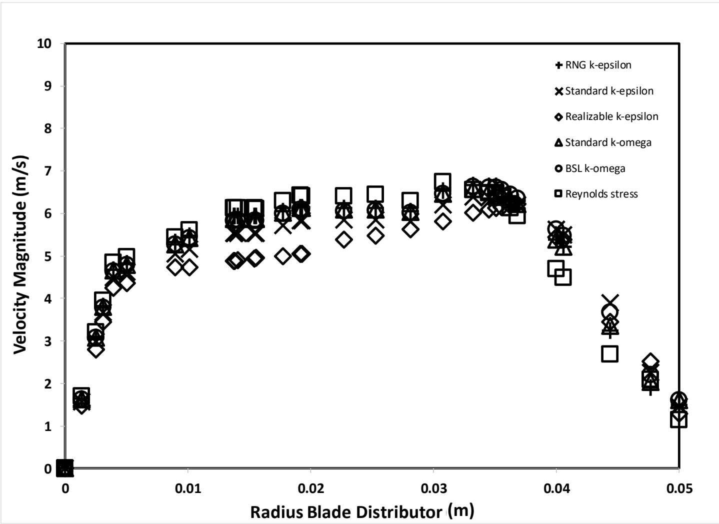


Fig. 5. Velocity distribution at various number of turbulence models

The flow is discernible in such a region. Significantly, the most important mean flow phenomena occur near to the centerline (e.g., vortex, breakdown, and recirculation zone). Consequently, the RNG $k{-}\varepsilon$ model is the most effective turbulent model for analysing the flow phenomena at the blade distributor, which is the topic of this study.

The RNG $k{-}\varepsilon$ model and the standard $k{-}\varepsilon$ model have comparable structures. In addition, the inclusion of the component in its ε equation would significantly improve the accuracy for flows subjected to fast strain and provide an analytical formula for turbulence Prandtl numbers. It is also applicable to the RNG model's swirl and turbulence condition [7,10].

In this current study, the RNG $k{-}\varepsilon$ model of Reynolds Averaged Navier Stokes (RANS) turbulence equation model is being considered for application. The RNG $k{-}\varepsilon$ model was the most prominent turbulence model used. Its dependability, numerical strength, and systematised forecasting capabilities have all been shown. For general-purpose simulations, the model follows the parameters from the earlier turbulence model research and provides a reasonable balance between accuracy and resilience [3,7].

3.3 Reynolds Number

When the twist blade distributor is oriented at a given angle, the Reynold's Number (Re) has a substantial effect on the flow characteristics through it. In the present design, Sreenivasan and Raghavan [1], Batcha and Raghavan [2], and Tawfik *et al.,* [7] performed a series of air flow investigations using a broad range of entering plenum chambers via a variety of blade numbers and blade inclination angles (10°, 12°, and 15°). As the flow approaches, there is a vertical component at the entrance of the trapezoidal space between the gap of two blade distributors. The direction of flow will be non-slippery.

The effects of Reynolds number, a simulation's raw data was also performed at several speeds of 2.25 m/s, 5.0 m/s, 6.75 m/s, 8.0 m/s, 18.0 m/s and 45.0 m/s respectively that corresponding to 14 881.8, 33 070.6, 44 645.29, 52 912.9, 119 054 and 297 635 Reynolds number, calculated from Eq. (5).

$$Re = \rho V_I D_H / \mu \quad (5)$$

where the dynamic viscosity, μ, density of air, ρ, hydraulic diameter plenum chamber ($D_H = 2(r_o - r_i)$) and velocity inlet, V_I were taken in accordance with the velocity inlet above. Therefore, this value showed an effect of fluid turbulence (Re > 4000) for all cases in the current work. As shown in Figure 6, the turbulence model provides a reading result that is quite similar to the RNG $k\text{−}\varepsilon$ turbulence model's.

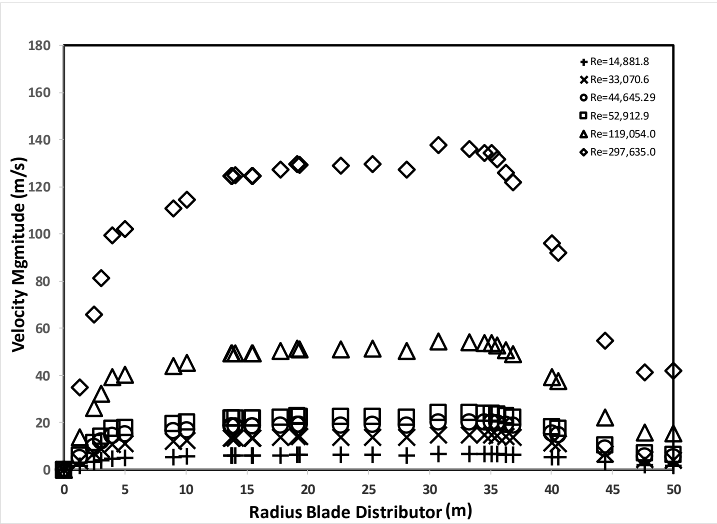


Fig. 6. Velocity distribution at various Reynold's numbers

3.4 Velocity Distribution Analysis

The current investigation contained 60 blades with constant horizontal inclination of 15 degrees, radial inclination of 10 degrees, and twist angle of 100 degrees. The geometry of the model is seen in Figure 3. The air will be deflected as soon as it reaches the distributor, producing a whirling effect. This will have an effect on the column's outer mass created by centrifugal force. Tangential velocity, radial velocity, and axial velocity are the three (3) components that form the velocity magnitude. In the actual industrial process, the effect on axial velocity induces fluidization while tangential velocity results in a swirling effect and radial velocity would be explained by the centrifugal force created by the rotating gas. In the next manuscripts, these velocity components will be addressed in more detail (Part 2 – The Air Flow Characteristics).

Consequently, the air flows are swirling due to the twist blade distributor. Additionally, the swirling action generates centrifugal force, which increases the mass of the air at the wall plenum edge, causing the flow to be separate into its three (3) component velocities. Figure 7 illustrates the gathered data where the number of blades, N_B, twist blade angle, T_B, blade radial inclination, I_R and blade horizontal inclination, I_H. The extracted data location of the velocity components was standardized in accordance to the previous study by Nawi [12]. In this section, validation of numerical simulation is presented. The validation is necessary to get agreement between numerical results of previous study. The simulation results of 60 blades with 15° blade inclination by Tawfik *et al.,* [7] were taken for validation. To optimize the current model of current fluidization systems the statistical analysis can be used in this study, the optimum configuration and parameter will be determined as previous study have been done [16-18].

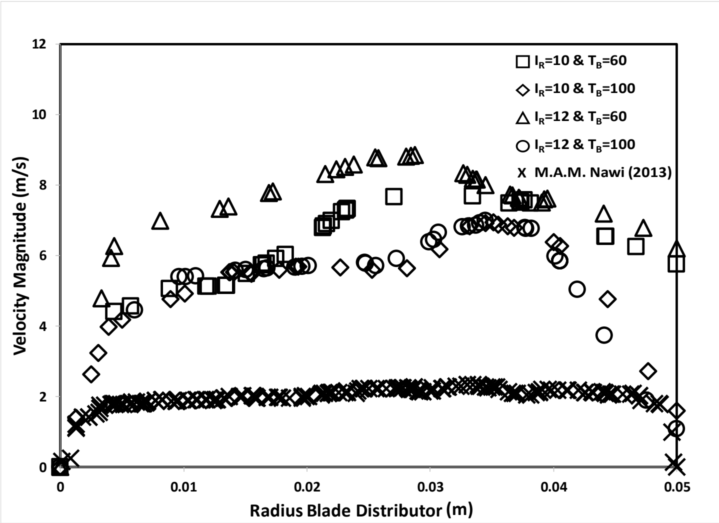


Fig. 7. Velocity magnitude for blade numbers, N_B = 60 with different twist blade angle, T_B = 60° and T_B = 100° via radial inclination, I_R = 10° and I_R = 12°

4. Conclusions

Numerical analysis is a useful tool for deciphering the complicated airflow phenomena in SFB, particularly in terms of velocity magnitude, which includes all three velocity components axial, tangential, and radial. This velocity magnitude reflects the annular blade distributor configuration's particular design, which allows for excellent mixing. A high air velocity magnitude will result in a high air tangential velocity. High tangential velocity will result in a better SFB system because it will have created the tornado effect, which can help speed up processes like drying. The tangential velocity of a distributor with a greater blade number is the highest.

Acknowledgement
The financial support provided by the Ministry of Higher Education Malaysia through Fundamental Research Grant Scheme reference no: FRGS/1/2018/TK02/UNIMAP/02/3 was acknowledged.

References
[1] Sreenivasan, Binod, and Vijay R. Raghavan. "Hydrodynamics of a swirling fluidised bed." *Chemical Engineering and Processing: Process Intensification* 41, no. 2 (2002): 99-106. https://doi.org/10.1016/S0255-2701(00)00155-0

[2] 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. https://doi.org/10.3923/jas.2011.1980.1986
[3] 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. https://doi.org/10.3923/ajsr.2013.157.166
[4] Mohideen, Mohd Faizal, Binod Sreenivasan, Shaharin Anwar Sulaiman, and Vijay Raj Raghavan. "Heat transfer in a swirling fluidized bed with geldart type-D particles." *Korean Journal of Chemical Engineering* 29, no. 7 (2012): 862-867. https://doi.org/10.1007/s11814-011-0255-6
[5] Tawfik, Mohamed Hamam M., Mohamed Refaat Diab, and Hamada Mohmed Abdelmotalib. "Heat transfer and bed dynamics study on a swirling fluidized bed under various inlet configurations." *International Journal of Thermal Sciences* 158 (2020): 106523. https://doi.org/10.1016/j.ijthermalsci.2020.106523
[6] Tawfik, Mohamed Hamam M., Mohamed Refaat Diab, and Hamada Mohmed Abdelmotalib. "An experimental investigation of wall-bed heat transfer and flow characteristics in a swirling fluidized bed reactor." *Applied Thermal Engineering* 155 (2019): 501-507. https://doi.org/10.1016/j.applthermaleng.2019.04.022
[7] Tawfik, Mohamed Hamam M., Mohamed Refaat Diab, and Hamada Mohmed Abdelmotalib. "Heat transfer and hydrodynamics of particles mixture in swirling fluidized bed." *International Journal of Thermal Sciences* 147 (2020): 106134. https://doi.org/10.1016/j.ijthermalsci.2019.106134
[8] Sitorus, Agustami, Satya Andika Putra, Irwin Syahri Cebro, and Ramayanty Bulan. "Modelling drying kinetics of paddy in swirling fluidized bed dryer." *Case Studies in Thermal Engineering* 28 (2021): 101572. https://doi.org/10.1016/j.csite.2021.101572
[9] Roslan, Nur Fatihah Mohd, Ahmmad Shukrie Md Yudin, Muhammad Afiq Sumono, Muhammad Faris Mohamad Pozi, and Muhammad Aidil Safuan Othman. "Drying Performance of Piper Nigrum in a Swirling Fluidized Bed Dryer: An Experimental Study." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 93, no. 1 (2022): 148-166. https://doi.org/10.37934/arfmts.93.1.148166
[10] Abd Latif, Muhammad Lutfi, Mohd Al Hafiz Mohd Nawi, Wan Azani Mustafa, Mohd Sharizan Md Sarip, Mohd Aminudin Jamlos, Masniezam Ahmad, Ku Mohammad Yazid Ku Ibrahim, and Hazizul Hussein. "Numerical analysis of velocity magnitude in a swirling fluidized bed with spiral blade distributor." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 59, no. 1 (2019): 38-44.
[11] Jamei, R., J. R. McDonough, D. A. Reay, and V. Zivkovic. "Rapid and intensified screening of a carbon capture adsorbent using a 3D-printed swirling fluidised bed." *Chemical Engineering Journal* 451 (2023): 138405. https://doi.org/10.1016/j.cej.2022.138405
[12] Nawi, Mohd Al-Hafiz Mohd. "Aerodynamics of a Swirling Fluidized Bed." *PhD diss., Universiti Tun Hussein Onn Malaysia*, 2013.
[13] ANSYS. *ANSYS Fluent Tutorial Guide Introduction Release 18.0*. Canonsburg, U.K., 2017.
[14] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education, 2007.
[15] Zulkifli, R. M., M. A. M. Nawi, M. I. Ishak, M. U. Rosli, S. N. A. Termizi, C. Y. Khor, and M. A. Faris. "Influence of Twisted Blades Distributor Towards Low Pressure Drop in Fluidization Systems." In *Intelligent Manufacturing and Mechatronics*, pp. 703-711. Springer, Singapore, 2021. https://doi.org/10.1007/978-981-16-0866-7_59
[16] Hazwan, M. H. M., Z. Shayfull, M. A. M. Nawi, M. Ahmad, Mohamad Syafiq AK, and A. M. Roslan. "Warpage optimization on battery cover using genetic algorithm (GA)." In *AIP Conference Proceedings*, vol. 2129, no. 1, p. 020195. AIP Publishing LLC, 2019. https://doi.org/10.1063/1.5118203
[17] Shayfull, Z., M. H. M. Hazwan, M. A. M. Nawi, M. Ahmad, Mohamad Syafiq AK, and A. M. Roslan. "Warpage optimization on battery cover using glowworm swarm optimisation (GSO)." In *AIP Conference Proceedings*, vol. 2129, no. 1, p. 020100. AIP Publishing LLC, 2019. https://doi.org/10.1063/1.5118108
[18] Ibrahim, M. I. F., M. U. Rosli, Muhammad Ikman Ishak, M. S. Zakaria, Mohd Riduan Jamalludin, C. Y. Khor, WM Faizal WA Rahim, M. A. M. Nawi, and Suhaimi Shahrin. "Simulation based optimization of injection molding parameter for meso-scale product of dental component fabrication using response surface methodology (RSM)." In *AIP Conference Proceedings*, vol. 2030, no. 1, p. 020078. AIP Publishing LLC, 2018. https://doi.org/10.1063/1.5066719