WEP Warwick Evans Publishing

International Journal of Mechanical and Electrical Engineering

ISSN: 3005-9615 (Print), ISSN: 3005-7132 (Online) | Volume 1, Number 1, Year 2023
DOI: https://doi.org/10.62051/ijmee.v1n1.10
Journal homepage: https://wepub.org/index.php/IJMEE/index

Study on circulating characteristics of circulating fluidized bed

Sijin Yu, Shaobei Liu

School of Mechanical Engineering, Sichuan University of Science & Engineering, Yibin 643000, China

ABSTRACT

Based on the Euler-Euler two-fluid model, the circulating fluidized bed was simulated using the computational fluid dynamics (CFD) method. The physical model of the circulating turbulent fluidized bed had an inner diameter of 76mm and a height of 2000mm. The circulating material consisted of class B particles, with an initial bed material mass (Mp) of 0.188kg. Air was uniformly introduced at a velocity of 0.3m/s at the bottom of the riser, and secondary air was introduced at a velocity of 0.9m/s at the secondary air inlet. Numerical simulations were conducted to study the gas-solid flow characteristics of the entire circuit. The turbulent bed, fast bed particle concentration, material circulation rate, and pressure drop gradient were analyzed to evaluate the performance of the entire circuit. The results showed that the particle concentration at the bottom of the riser was higher in the overall circuit flow, and a turbulent state could be achieved at low gas velocities. The particle concentration in the transport bed was low, indicating good gas-solid interaction. Strong backmixing occurred at the connection between the return valve and the turbulent bed. Additionally, there was intense gas-solid reaction at the secondary air inlet.

KEYWORDS

Circulating fluidized bed; Two-phase flow; Numerical simulation.

1. INTRODUCTION

The flow characteristics are an important research object of cyclic fluidized bed reaction equipment, and its main parameters include: particle circulation flow rate Gs, bed pressure drop ΔP, etc. The structure of the equipment and the characteristics of the solid particles have an important influence on the operating parameters. If the parameter Gs is too high, it may accelerate the loss of equipment components, and the burden on blowers and separators will increase. However, if the parameter Gs is too low, the heat and mass transfer capacity can be reduced, and the reaction space will be consumed. Therefore, the flow characteristics are one of the key characteristics of CFB, and the study of this parameter can be used to adjust and optimize the equipment. Provides support for the amplification of the device.

The general circulating flow rate is measured by the measurement of particles per unit of time. However, due to the dimensional accuracy of the equipment structure, the amount of particles, and the wear and tear during operation, it is impossible to have exactly the same conditions in each experiment. At the same time, the accuracy of the measuring instruments is limited, and the confidence in the data obtained from these experiments may be reduced to reflect the detailed information inside the experiment. The above multiple factors inevitably lead to differences in experimental measurements, so a higher fidelity method is needed to study the phenomenon of particle distribution, pressure changes. Computational fluid dynamics (CFD) is thus used as an alternative as a powerful tool for modeling multiphase flows, suitable for studying gas-solid flow dynamics and non-uniform distribution phenomena in CFBs with cyclones and other systems.

Chen et al. [2] used the EMMS drag model to simulate the mixing and flow of gas-solid two-phase flow in the feed mixing section of riser. Zhang et al. [3] used the discrete element coupling method to numerically simulate the flow characteristics of FCC particles in the riser tube of the circulating turbulent fluidized bed. Theeranan Thummakul et al. [4] simulated the turbulent flow of gas-solids in the lifting tube and the particle exchange capacity in the tube. In general, the simulations of the researchers only study the flow behavior of the particles in the riser, and the simulations show that the results can satisfy the axial symmetry of the turbulence [5]. However, there are few studies on the cyclic turbulent flow of gas-solid as a whole. Therefore, the author uses the particle dynamics Euler-Euler two-fluid model and SSTk-ω model to simulate the complete turbulent process of the gas phase (wall-bound flow and free shear flow), and numerically simulates the whole system of this new type of cyclic turbulent fluidized bed, focusing on the analysis of the turbulent flow characteristics of the cyclic dense phase region, and discusses the cyclic turbulent fluidization. Provides support for the optimization of circulating turbulent fluidized beds。

2. MATHEMATICAL MODELS

Apply numerical simulations and multiphase flow mechanistic methods provided by computational fluid dynamics (CFD). Prediction of fluid dynamics based on Euler-Euler fluid model [27]. The gas-phase and solid-phase interactions were simulated using the software Fluent 2022R1. In the model, according to the standard kinetic theory of particle flow (KTGF), the dispersed phase such as bubbles and the gas phase are regarded as quasi-continuous media, and the bubbles and gas are common and independent quasi-fluids, which can penetrate each other, while in the Eulerian coordinate system, a controlled conservation equation similar to that of continuous fluids is required to represent the gas and solid phases respectively. The equipment in this paper is a circulating turbulent fluidized bed, including a dense phase turbulent flow stage, a pneumatic conveying stage and a particle separation stage. Each stage contains a variety of gas-solid effects, and the flow state is complex. It is difficult for a single dense phase gas-solid drag model to accurately reflect the real behavior of gas-solid fluidization in a fluidized bed. In the simulation, the Gidassow gas-solid drag model was used for the drag model [7]. The Gidaspow model is a combination of the Wen&Yu model and the Ergun equation and is suitable for dense phase gas-solid fluidized beds.

3. SIMULATION MODEL

Figure 1 is a schematic diagram of the 3D modeling of the CFB reactor. The whole circuit mainly consists of five parts: turbulent bed, fast bed, separator, storage tank, and return valve.

The particles were set to particles with a particle size of 100 μm and a particle density of 1500 kg/m3, which belonged to Geldart class B particles, and the turbulent flow of particles was significant. It is convenient to simulate particle flow behavior. The volume fraction of solids stacked in the lower part of the turbulent bed is 0.5, the initial stationary accumulation height is 1.2m, and the particle collision recovery coefficient is 0.95. Different discretization schemes are used for the convection terms of each governing equation: the momentum equation adopts the second-order headwind scheme; The volume fraction of the gas and solid phases was selected for the convective kinematics (QUICK) scheme, and the first-order headwind scheme was chosen for the turbulent energy and turbulent dissipation rate. The pressure correlation equation semi-implicit (SIMPLE) algorithm is used to deal with the pressure-velocity coupling problem. The time step is assigned to 0.00001 seconds, and the total simulation time of the calculation step is 7 seconds. As shown in Figure 1(b), in order to be close to the actual flow of the model, a hexahedral structured mesh is used, and the reactor informs the walls.

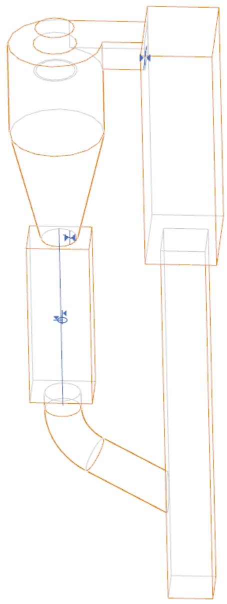

Fig.1 Schematic diagram of the overall model

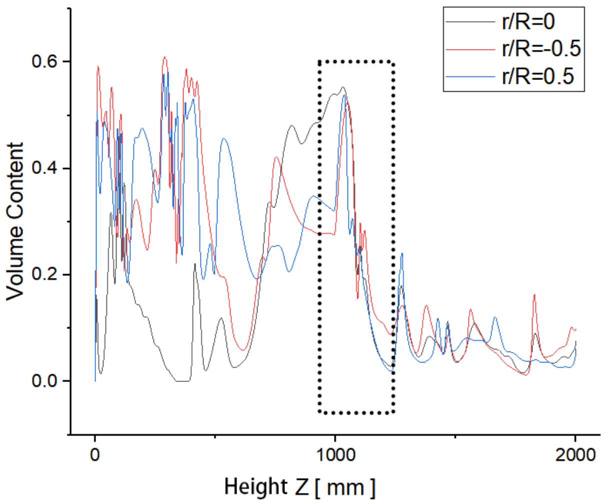

Fig.2 Axial solid content plot of particles

3.1. Concentration distribution

The axial distribution of particle volume content is shown in Figure 2. At the height of 600mm to 1200m in the axial direction, the particle volume content was mostly distributed between 0.1 and 0.5, and the average particle volume content of the three points in the central region (r/ R = -0.5, 0, 0.5) were 0.28, 0.31 and 0.41, respectively. The particle volume content is distributed irregularly along the axial trigonometric function, indicating that the gap between the solid particle phase and the gas phase is flocculent, and the particle volume content in the axial center region is approximately uniform, which is consistent with the particle volume content distribution of the pressure change reaction.

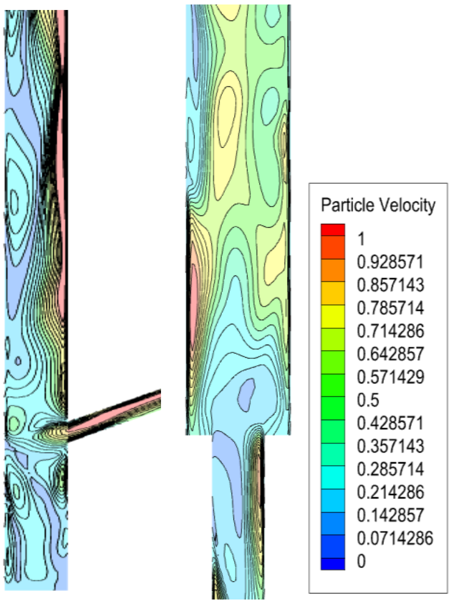

Fig.3 Particle velocity contour

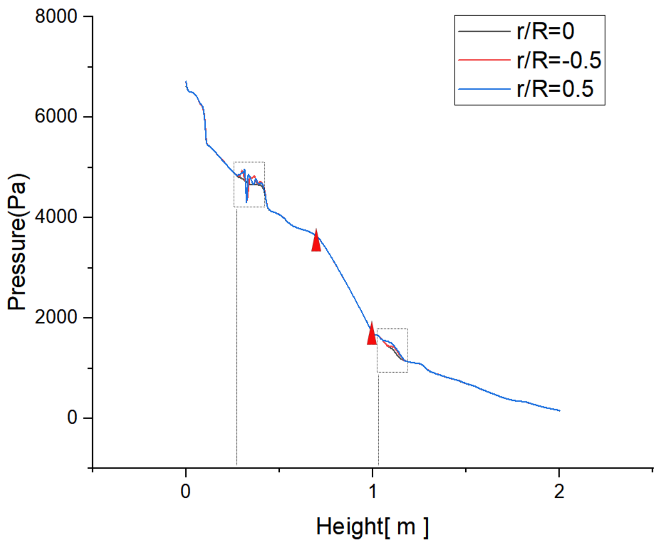

Fig.4 Pressure distribution diagram

3.2. Pressure

In the wireframe on the left side of Figure 4, it can be seen that the cross-sectional pressure fluctuates, and after the particles are returned to the turbulent bed by the return valve, the effect of the return material may cause the local particle pressure to rise, forming the Kangda effect. Clusters of particle particles cause uneven distribution of solids in turbulence, which strongly causes pressure fluctuations. The pressure fluctuation in the wireframe on the right side of Figure 4 is due to the jet effect at the inlet of the secondary wind, as shown in Figure 5, the slope of the marking section at a height of 1 m is larger, and the pressure change is more drastic than that of the turbulent section, forming a certain pressure fluctuation in the inlet plane. Under the action of high gas velocity, the particle movement is intensified, which is conducive to particle transport. The particles showed a distribution state of sparse at the top and concentrated at the bottom, and the particle supply was insufficient, and the upper part was formed as a sparse phase segment. The posterior pressure tends to a stable state. The ingress of secondary air facilitates the efficiency of gas delivery as well as the mixing of particles.

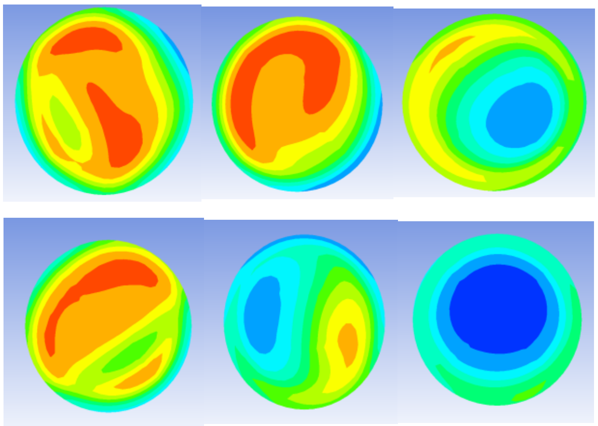

Fig.5 Particle volume distribution contour

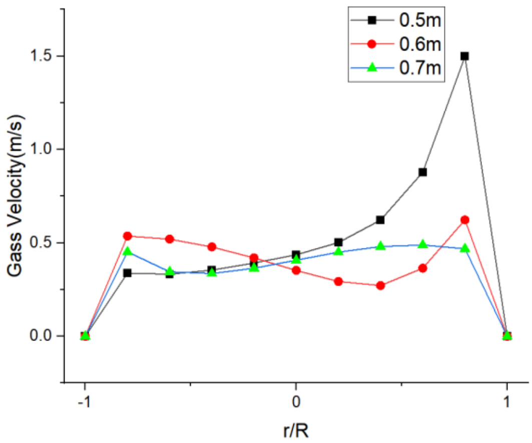

Fig.6 Radial distribution of gas phase velocity

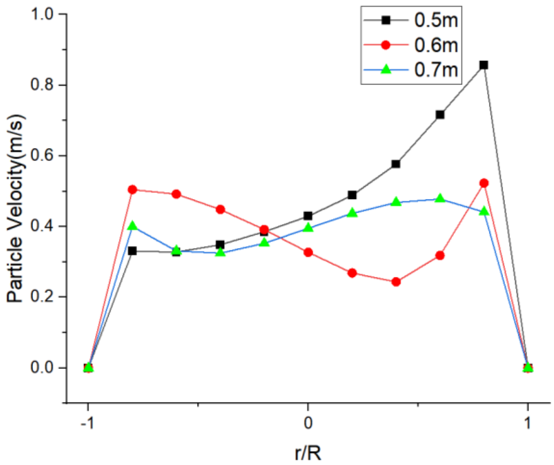

Fig.7 Radial distribution of gas phase velocity

3.3. Particle velocity distribution

As shown in Fig. 6 and 7, it shows that the velocity of the solid phase in the center is high, and the wall velocity is approximately 0 m/s, and there is a phenomenon of repeated mixed flow of gas and solids at the side wall due to the stagnation of particles. The axial velocity of the particle phase is obviously different at different heights, and when h=0.5 and 0.7m, the particle velocity crosses with that at h=0.6m, indicating that the particles reciprocate with the increase of height in the area of 0.5~0.7m.

4. CONCLUSION

In this paper, a multi-fluid model based on the Euler-Euler framework is used to numerically study the characteristics of flow phenomena in a circulating turbulent fluidized bed. The cycling characteristics of CFB in the full loop are discussed. Based on the numerical results, the following main conclusions can be drawn:

1)Under the condition of Class B particles, the circulating turbulent bed requires less wind speed than the general turbulent bed to reach the turbulent state (usually 1.5 m/s). The turbulent section does not produce significant large bubbles；

2)The axial pressure of the turbulent section and the conveying section varies linearly with the height, especially in the turbulent section, the particle volume content is as high as 0.2 or more and fluctuates, and the particle volume content on the wall surface is high, and there is a strong gas-solid backmix, which increases the particle residence time and is conducive to particle reaction.

3)The particles will reciprocate during the movement, and this phenomenon can be used to aggravate the gas-solid reaction.

4)In the circulating turbulent fluidized bed, gas-solid backmixing is an unavoidable phenomenon.

REFERENCES

[1] Theeranan Thummakul, Dimitri Gidaspow,et al. CFD simulation of CO2 sorption on K2CO3 solid sorbent in novel high flux circulating-turbulent fluidized bed riser: Parametric statistical experimental design study.[J]: Applied Energy 2017(190):122–134.

[2] Jesse Zhu. Circulating turbulent fluidization—A new fluidization regime or just a transitional phenomenon.[J]: Particuology (2010) 640–644.

[3] Min wang. Numerical Simulation of the Pilot-scale High-Density Circulating fluidized Bed Rizer.[J]: industrial&Engineering Chemical Research 2021,60,3184-3197.

[4] Shuyue Li,Yansog ShenCFD investigation of maldistribution in a full-loop circulating fluidized bed with double parallel cyclones[J] Powder Technology 381(2021)666-684

[5] HANES D, INMAN D. Observations of rapid flowing granular-fluid flow[J]. Journal of Fluid Mechanics, 1985, 150: 357-380.

[6] KRISHNA R, VAN BATEN J M, ELLENBERGER J. Scale effects in fluidized multiphase reactors[J]. Powder Technology, 1998, 100(2): 137-146.

[7] JOHNSON P, JACKSON R. Frictional-collisional constitutive relations for granular materials, with application to plane shearing[J]. Journal of Fluid Mechanics, 1987, 176: 67-93.