Numerical and experimental investigation of aerodynamics and solid particle dynamics in a diffuser with fluidized and fountaining beds

Rakhimjan Babakhodjaev[1], Dilnoza Pulatova [1, 2, a)], Charos Pulatova[1]

[1] *Tashkent state technical university named after Islam Karimov, Tashkent, Uzbekistan*
[2] *Karakalpak State University named after Berdakh, Nukus, Karakalpakstan, Uzbekistan*

[a)] *Corresponding author: pulatovadilnoza83@gmail.com*

Abstract This study presents a combined numerical and experimental investigation of airflow and solid particle dynamics within a diffuser containing fluidized and fountain layers. The research focuses on the influence of local pressure drop, diffuser geometry, and phase interactions on the velocity distribution and turbulence characteristics of the two-phase flow. Experimental measurements were conducted using a single-chamber setup with sunflower seed particles as the model material, while numerical simulations employed Euler-Lagrange and CFD-DEM approaches. Results demonstrate that velocity profiles vary significantly with pressure drop and across the diffuser cross-section. The onset of the fountain regime leads to a marked increase in turbulence intensity and complex vortex structures, enhancing velocity fluctuations in both axial and radial directions. Numerical results further confirm a 20–35% rise in turbulent kinetic energy in the upper diffuser region compared to the fluidized state. These findings highlight the critical role of diffuser design, pressure gradients, and operational regimes in optimizing heat and mass transfer processes in industrial systems. The study provides valuable insights for improving phase flow homogenization and stability in multiphase apparatuses. The Ghost thorough scheme experimental installation. Using new instruments for determination of the velocities mixture and pressures. The Graphic determination to profiles to velocities in each section on height 10 refer to specified to installation. Specify dependency event fountain of the hard particles of model backfilling from intensity of the turbulences of the flow.

INTRODUCTION

Currently, the study of air and solid particle flow in systems with fluidized and fountaining beds is gaining increasing relevance due to the widespread application of such technologies in the chemical, energy, and processing industries. One of the key components of these systems is the diffuser, where complex hydrodynamic phenomena occur, significantly influencing the efficiency of heat and mass transfer processes. Despite the large number of studies devoted to fluidization, the detailed flow structure in the diffuser zone remains insufficiently investigated, particularly under transitional regimes between fluidization and fountaining.

The present study is focused on the numerical and experimental investigation of airflow characteristics and solid particle behavior within a diffuser containing fluidized and fountaining beds. The obtained results are expected to provide a deeper understanding of the interphase interaction mechanisms and contribute to the optimization of diffuser design parameters in engineering applications.

The historical analysis of scientific progress demonstrates that advancements in the field of channel heat transfer mechanics and the intensification of Angren coal combustion have consistently been driven and significantly influenced by achievements in experimental research. It is important to emphasize the critical role of hydrodynamic and aerodynamic experimentation in the development and optimization of various burner systems, boiler units, and their individual components [1.3].

The primary concerns for engineers when analyzing the flow of liquids and gases in pipes are the velocity distribution and pressure drop across the pipe's cross-section. Experimental studies have shown that both velocity profiles and pressure losses can vary significantly depending on factors such as pipe diameter, fluid velocity and viscosity, pipe wall roughness, as well as the concentration of solid impurities in liquids and gases.

The experiments were conducted using a single-chamber experimental setup with an intensified fluidized bed (IFB) developed by our team [2]. A fixed bed (FB) was formed in this setup for model studies. Sunflower seeds, roasted prior to use, served as the solid particles. The height of the fixed bed varied between 10 and 25 cm.

The fixed bed consists of a network of interconnected and tortuous channels with varying cross-sections. Instantaneous velocities of liquid and gas flow within this layer can reach relatively high values.

Gas flow within the fixed bed differs significantly from conventional turbulent flow, as the flow in the fixed bed under these conditions is a complex, unsteady, pulsating regime. It consists of relatively high-velocity jets penetrating into adjacent segments within the diffuser of the fixed bed, where they become chaotic. Thus, the liquid or gas flow in the apparatus's fixed bed represents a flow with a structure that continuously changes the motion of the liquid and gas [4].

The Euler–Lagrange method is used for numerical modeling with the application of CFD packages (e.g., ANSYS Fluent), where the gas phase is described as a continuous medium, and the solid particles are tracked as discrete elements. Turbulence is modeled using the k-ε turbulence model, and the interaction between phases is implemented through drag force models such as the Drag or Gidaspow models [5, 6, 7].

EXPERIMENTAL RESEARCH

The experimental section involves a setup with a transparent diffuser equipped with a visualization system (e.g., PIV or high-speed video recording) and a pressure measurement system. Glass beads are used as the solid particle simulant material, and the flow velocity is varied within the transition range between flow regimes.

Experimental studies and visual observations of the flow of mixtures in pipelines and channels provide a basis for selecting a hydrodynamic model when investigating the combustion theory of finely dispersed solid particle mixtures. Due to the absence of a unified theory for all types of combustion, separate hydrodynamic models are applied, such as the quasi-homogeneous model, the interpenetrating model, heterogeneous (multiphase) models, and others. The flow of mixtures is considered as a quasi-homogeneous medium with averaged parameters and is described by equations whose parameters are partially calculated using expressions that are not universal for all types of mixtures but are specific to particular combustion cases. For example, these models are applied to describe the motion of dispersed particles of Angren coal.

The aim of this study is the experimental determination of the variation in the averaged velocity of a two-phase flow along the length of the diffuser as a function of pressure drop and flow cross-sections.

To investigate the flow structure of certain mixtures in a vertical diffuser pipe, a single-chamber experimental setup with an intensified fluidized bed (IFB) was developed.

a) For visual observation and photographing the structure of vertical flow formation.

b) For calculating the distribution and characteristics of solid particle concentration across the diffuser cross-section.

c) For measuring and determining the nature of the averaged velocity profiles across the diffuser cross-section.

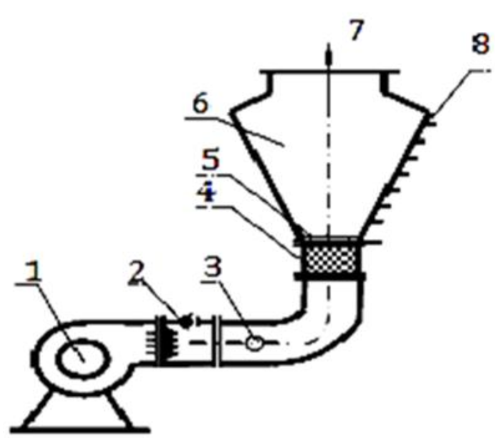

FIGURE 1. Schematic diagram of the experimental setup.
1 - Fan; 2 - Airflow regulator; 3, 8 - Ports for installing measuring sensors; 4 - Airflow stabilizer; 5 - Retaining grid; 6 - Working chamber; 7 - Air outlet.

The setup consists of a conical working chamber (6) made of acrylic glass, allowing visual observation of the hydrodynamics of the process (Fig. 1). Vertical pressure taps (8) are installed on the chamber body (6) at 10 cm intervals for measuring pressure drop. The lowest tap is located at the gas flow inlet to the chamber. An airflow stabilizer is placed in the inlet duct of the air supply pipe to smooth the swirling gas flow before entering the working chamber. The diameter of the inlet opening is 10 cm. Air is supplied to the chamber by a fan (1), and its flow rate is regulated by a special flow regulator (2). Measuring sensors are installed in the ports (3 and 8). The control and measurement instruments used include a Testo 405-V1 flow meter, a differential micromanometer MMN-2400, and a digital video camera GR-D850AS.

As a model material, a mixture consisting of sunflower seed components was used (seeds - 80%, husk - 10%, and pulp - 10%). The methodologies for flow velocity measurement are described in references [2,8].

RESEARCH RESULTS

In our previous works, the Reynolds number correlation was obtained using the formula [9].

The first stage of the experiment involved measuring the velocity fields across the cross-sections of the diffuser with a fixed bed (sunflower seeds). The average air flow velocities were measured using the modern device Testo 405-V1. The pressure drop was measured with the MNH-2400 instrument [12.13]. Velocity fields were measured separately at each cross-section. The obtained results, after statistical processing of the average velocities up to the midpoint of the setup, are presented in Figures 2.

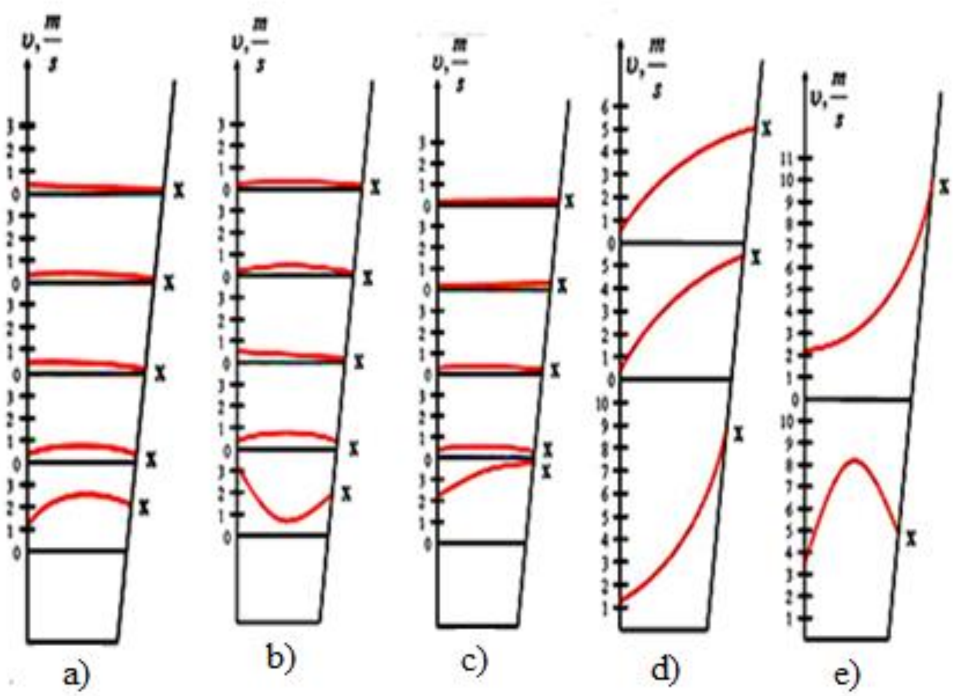

FIGURE 2. Graphical dependencies of the velocity profile at 10 cm model bed height.
a) ΔP = 20 Pa; b) ΔP = 30 Pa; c) ΔP = 45–50 Pa; d) ΔP = 60–70 Pa;
e) ΔP = 60–80 Pa (fountain regime).

From Fig. 2, it is evident that the velocity profiles depend on the pressure drop and vary across the cross-section of the apparatus (diffuser). After the onset of the fountain regime, the turbulence intensity of the solid particles increases significantly.

CONCLUSIONS

Based on numerical and experimental analysis, it has been established that the velocity distribution of the gas phase and solid particles in the diffuser significantly depends on the local pressure drop, which arises due to changes in the apparatus geometry and phase interactions in the zones of the fluidized and fountain layers. The increase in the diffuser cross-sectional area leads to changes in the pressure and velocity fields, resulting in characteristic velocity gradients across the width of the cross-section, as well as secondary flows that promote particle mixing.

A significant increase in turbulence intensity is observed, which is associated with the high-energy ejection of solid particles upward and their subsequent interaction with the counter-flowing gas stream. This leads to the formation of complex vortex structures that enhance velocity fluctuations in both axial and radial directions. Similar effects are described in the works of Zhou et al. (2017) and Mahajan et al. (2018), where it is also noted that the transition to the fountain regime is accompanied by a sharp increase in velocity dispersion and local flow instability [10,11].

Additionally, numerical calculations using Euler-Lagrange and CFD-DEM models confirmed that the turbulent kinetic energy (TKE) increases by 20–35% in the upper part of the diffuser compared to the fluidized regime. This effect is crucial for optimizing heat and mass transfer processes in industrial apparatuses employing diffusers for phase flow homogenization and stabilization.

Thus, the diffuser geometry, pressure drop, and operating regime (fluidization or fountain) play a critical role in shaping the aerodynamic characteristics of the flow and must be taken into account during the design and scaling of such systems.

REFERENCES

1. Babakhodjaev R. P., "The two-phase gushing forth layer in the conic device," *European Applied Sciences*, No. 3, 96–100 (2014).
2. Babakhodjaev R. P., Karimov A. A., and Shakirov A. A., "Hydrodynamic studies of a two-phase gushing fluidized layer in a conical apparatus," *Vestnik Tashkent State Technical University*, No. 3–4, 79–82 (2009).
3. Babakhodjaev R. P. *et al.*, "Intensified fluidized bed for combustion of Angren brown coal with increased ash content," in *Proc. Int. Scientific Seminar NATO–Kyiv* (2006), pp. 46–55.
4. Adamczyk W. P., Węcel G., Klajny M. *et al.*, "Modeling of particle transport and combustion phenomena in a large-scale circulating fluidized bed boiler using a hybrid Euler–Lagrange approach," *Particuology* **16**, 129–140 (2014).
5. Alobaid F., Ohlemüller P., Ströhle J. *et al.*, "Extended Euler–Euler model for the simulation of a 1 MWth chemical-looping pilot plant," *Energy* **93**, 2395–2405 (2015).
6. Banerjee S. and Agarwal R. K., "An Eulerian approach to CFD simulation of a chemical-looping combustion reactor with chemical reactions," *J. Phys. Chem. B* **110**, 4756–4765 (2015).
7. Adamczyk W. P., Klimanek A., and Białecki R. A. *et al.*, "Comparison of the standard Euler–Euler and hybrid Euler–Lagrange approaches for modeling particle transport in a pilot-scale circulating fluidized bed," *Particuology* **15**, 129–137 (2014).
8. Babakhodjaev R. P., Mukhiddinov D. N., Karimov A. A., Shakirov A. O., Khodjaev B. A., Yusupov B. V., Pulatova D. M., and Khujanov R. A., *Installation for Burning Solid Fuels or Heat Treatment of Polydisperse Materials*, Patent No. IAP 04840 (Intellectual Property Agency of the Republic of Uzbekistan, Official Bulletin No. 3, 2014), p. 49. Available: http://baza.ima.uz/upload/Bulletin/2014/03%20(155)%2031-03-2014/bul3-2014.pdf
9. Pulatova D. M., "Analysis of hydrodynamic processes in a fluidized bed," *Problems of Energy and Sources Saving*, No. 2, 69–73 (2025). DOI: 10.5281/zenodo.15743
10. Zhou Y., Wang W., and Jin Y., "Experimental study and transient CFD/DEM simulation in a fluidized bed based on different drag models," *RSC Advances* **7**, 53067–53077 (2017). DOI: 10.1039/C6RA28615A
11. Mahajan D. and Dudukovic M. P., "Nonspherical particles in a pseudo–2D fluidized bed: Experimental study," *AIChE Journal* **64**(8), 2933–2944 (2018). DOI: 10.1002/aic.16078
12. Babakhodjaev R., Pulatova D., Mirzaev J., and Kurbanbaeva Z., "Aerodynamics of the solid particle in the ascending stream of the intensified fluidized bed," *AIP Conference Proceedings* **2552**, 050025 (2023). DOI: 10.1063/5.0112010
13. Pulatova D., Pulatov B., and Pulatova C., "Rotating regenerative heat recovery units in ventilation systems," *AIP Conference Proceedings* **3152**, 040011 (2024). DOI: 10.1063/5.0219599