3-бөлім Раздел 3 Section 3

Механика Механика Mechanics

IRSTI 30.17.35; 27.35.45; 44.31.00 DOI: https://doi.org/10.26577/JMMCS202512849

N. Momysh[1,2], Ye. Belyayev[1*], O. Botella[2]
[1] Al-Farabi Kazakh National University, Almaty, Kazakhstan
[2] Université de Lorraine, CNRS, LEMTA, Nancy, France
e-mail*: yerzhan.belyaev@kaznu.edu.kz

NUMERICAL ANALYSIS OF FLUIDIZED BED HYDRODYNAMICS WITH OPENFOAM

Gas–solid fluidized beds play a vital role in energy production, chemical processing, and thermal management due to their excellent mixing and transport properties. Despite their importance, predicting fluidized bed hydrodynamics remains a major challenge because of the highly coupled and nonlinear interactions between gas and particle phases. Computational fluid dynamics (CFD) has become an indispensable tool for analyzing such systems, but the reliability of predictions depends strongly on solver formulation, closure models, and postprocessing strategies. This study revisits the benchmark experiment of Taghipour et al. [1], which provides high-quality measurements of pressure drop and bed expansion BER, and applies it to the most recent release of OpenFOAM (v12). An Euler–Euler two-fluid approach is employed, incorporating kinetic theory of granular flow for solid-phase stresses and the Gidaspow drag correlation for interphase momentum exchange. Simulations are performed on a two-dimensional rectangular bed fluidized with air and Geldart B particles. Pressure drop and bed expansion ratio (BER) are selected as the main indicators for validation. Beyond conventional postprocessing methods, a new mass-conservation-based approach for estimating BER is introduced, which takes into account data from the entire computational domain. The work aims to evaluate the predictive capacity of OpenFOAM v12 in reproducing well-established benchmarks and to advance postprocessing techniques for more reliable characterization of fluidized bed hydrodynamics.

Key words: Fluidized bed, OpenFOAM, two-phase model, postprocessing methods, pressure drop, layer expansion coefficient.

Н. Момыш[1,2], Е. Беляев[1*], О. Ботелла[2]
[1] Әл-Фараби атындағы Қазақ ұлттық университеті, Алматы, Қазақстан
[2] Лотарингия Университеті, CNSR, LEMTA, Нанси, Франция
e-mail*: yerzhan.belyaev@kaznu.edu.kz

OpenFOAM көмегімен сұйытылған қабат гидродинамикасының сандық талдауы

Газ–қатты фазалы сұйытылған қабаттар энергия өндіру, химиялық технологиялар және жылуалмасу процестерінде тиімді араластыру және массаны/жылуды тасымалдау қабілеттеріне байланысты кеңінен қолданылады. Алайда мұндай жүйелердегі гидродинамикалық құбылыстарды дәл болжау газ және қатты бөлшектер фазалары арасындағы күрделі әрі сызықты емес байланысқандықтан әлі күнге дейін күрделі мәселе. Қазіргі таңда CFD сұйытылған қабаттарды зерттеудің негізгі құралдарының бірі, бірақ модельдеу нәтижелерінің сенімділігі шешуші теңдеулердің нұсқасына, қолданылған жабу модельдеріне және деректерді өңдеу әдістеріне тікелей тәуелді. Осы зерттеуде Taghipour et al. [1] қысым түсуі мен қабаттың ұлғаюы жөніндегі жоғары дәлдіктегі эксперименттік деректері қарастырылып, OpenFOAM бағдарламасының соңғы нұсқасында (v12) сандық талдау жүргізіледі. Екі фазалы Эйлер–Эйлер моделі қолданылып, қатты фаза кернеулері гранулалық ағынның кинетикалық теориясымен, ал фазалар арасындағы импульс алмасу Gidaspow тарту корреляциясымен сипатталады. Сандық модельдеу ауа ағынымен сұйытылған Geldart B санатына жататын бөлшектері бар екі өлшемді тікбұрышты қабат үшін орындалды. Негізгі валидациялық көрсеткіштер ре-

тінде қысым түсуі және қабаттың ұлғаю коэффициенті (BER) алынған. Бұған қоса, есептеу ауданындағы деректерді толық ескере алатын, BER-ді масса сақталуы принципіне негізделген жаңа есептеу тәсілі ұсынылады. Зерттеу нәтижелері OpenFOAM v12 нұсқасының жақсы белгілі эталондық гидродинамикалық сипаттамаларды қайта өндіру қабілетін бағалауға және сұйытылған қабаттарды талдауға арналған деректерді өңдеу әдістерін жетілдіруге бағытталған.
Түйін сөздер: Сұйытылған қабат, OpenFOAM, екі-фазалы модель, деректерді өңдеу, қысымның түсуі, қабаттың ұлғаю коэффициенті.

Н. Момыш[1,2], Е. Беляев[1*], О. Ботелла[2]
[1] Казахский национальный университет имени Аль-Фараби, Алматы, Казахстанх
[2] Университет Лотарингии, CNRS, LEMTA, Нанси, Франция
e-mail*: yerzhan.belyaev@kaznu.edu.kz

Численный анализ гидродинамики псевдоожиженного слоя с использованием OpenFOAM

Газо–твердые псевдоожиженные слои широко применяются в энергетике, химической промышленности и теплотехнике благодаря высоким характеристикам смешения, тепло- и массообмена. Тем не менее, точное прогнозирование гидродинамики таких систем остаётся сложной задачей из-за интенсивных, нелинейных и тесно связанных взаимодействий газовой и твёрдой фаз. Вычислительная гидродинамика (CFD) сегодня является ключевым инструментом для анализа псевдоожиженных систем, однако достоверность численного моделирования в значительной степени зависит от выбора численного решателя, моделей замыкания и применяемых методов постпроцессинга. В настоящей работе повторно рассмотрены высокоточные экспериментальные данные по падению давления и расширению слоя, представленные Taghipour и соавторами [1], и проведено их численное воспроизведение в последней версии OpenFOAM (v12). Используется двухфазный Эйлер–Эйлеровский подход, где напряжения твёрдой фазы описываются кинетической теорией гранулярного потока, а межфазный обмен импульсом — корреляцией сопротивления Gidaspow. Моделирование выполнено для двумерного прямоугольного псевдоожиженного слоя, аэрируемого воздухом и содержащего частицы типа Geldart B. В качестве основных критериев валидации приняты падение давления и коэффициент расширения слоя (BER). Кроме того, предложен новый метод оценки BER, основанный на законе сохранения массы, который позволяет учитывать данные всей вычислительной области. Результаты исследования направлены на оценку способности OpenFOAM v12 воспроизводить общепринятые эталонные характеристики гидродинамики псевдоожиженного слоя и на совершенствование методов постпроцессинга для более надёжного анализа таких систем.
Ключевые слова: Псевдоожиженный слой, OpenFOAM, двухфазная модель, постпроцессинг, падение давления, коэффициент расширения слоя.

1 Introduction

Fluidized beds are widely applied in energy conversion, chemical processing, and waste heat recovery due to their excellent mixing and transport properties [2]. However, predicting their hydrodynamic behavior remains challenging because of the complex interactions between gas and solid phases. Computational fluid dynamics (CFD) has become an essential tool for analyzing these systems, but numerical predictions often diverge from experiments, making reliable benchmarks crucial for model validation. In a related context, recent CFD studies of thermal energy storage systems have demonstrated that two-dimensional numerical models, validated against experimental data, can accurately capture complex flow structures and performance indicators such as temperature stratification, efficiency metrics, and mixing behavior, highlighting the broader applicability of CFD methodologies beyond fluidized beds [3]. One of the most widely used benchmarks is the experiment of Taghipour et al. [1], which provides high-quality measurements of global flow quantities such as pressure drop and bed expansion ratio (hereafter BER) in a two-dimensional gas–solid fluidized bed. These data have been used extensively to test Euler–Euler two-fluid models (TFM) combined with drag correlations such as Wen–Yu [4], Syamlal–O'Brien [5], and Gidaspow [6]. While many studies have reproduced this benchmark, results remain sensitive to choices of drag law, boundary conditions. Previous evaluations of OpenFOAM have shown mixed conclusions: some reported insufficient accuracy, while others demonstrated close agreement with experiments. Importantly, no study has yet assessed the most recent release, OpenFOAM v12, which incorporates updates in solver robustness and numerical schemes. Addressing this gap, the present work investigates whether the multiPhaseEuler solver in OpenFOAM v12 can reliably reproduce Taghipour's [1] benchmark. Pressure drop and BER are selected as the main hydrodynamic indicators. In addition, a new mass-conservation-based method for calculating BER is proposed and compared with conventional approaches. The objective of this study is to validate the predictive capability of OpenFOAM v12 for gas–solid fluidized bed hydrodynamics and to provide improved postprocessing strategies that can support industrial-scale applications.

2 Materials and Methods

This study reproduces the benchmark experiment of Taghipour et al. [1] using the Euler–Euler two-fluid model in OpenFOAM v12. The objective is to assess the solver's ability to predict two key hydrodynamic indicators: pressure drop and bed expansion ratio (BER). The simulated system is a two-dimensional rectangular bed with dimensions of 1.0 m $\times$ 0.28 m $\times$ 0.025 m. The bed is initially filled to 40% of its height with Geldart B particles (mean diameter 275 μm, density 2500 kg/m^3). Air serves as the fluidizing medium, with density 1.225 $\mathrm{kg\,m^{-3}}$ and kinematic viscosity $1.485 \times 10^{-5}\,\mathrm{m^2\,s^{-1}}$. The superficial gas velocity was varied between 0.025 and 0.51 m/s, covering the transition from fixed to bubbling and turbulent regimes. The simulation utilized a multifluid Eulerian approach that solves conservation equations for mass and momentum across gas and fluid phases. For modeling solid-phase stresses, the kinetic theory of granular flow was implemented, providing closure through conservation of solid fluctuation energy [7]. The governing equations can be summarized as follows:

Mass conservation equations of gas (g) and solid (s) phases:

$$\frac{\partial}{\partial t}(\alpha_g \rho_g) + \nabla \cdot (\alpha_g \rho_g \vec{v}_g) = 0, \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_s \rho_s) + \nabla \cdot (\alpha_s \rho_s \vec{v}_s) = 0. \tag{2}$$

Momentum conservation equations of gas (g) and solid (s) phases:

$$\frac{\partial}{\partial t}\big(\alpha_g \rho_g \vec{u}_g\big) + \nabla \cdot \big(\alpha_g \rho_g \vec{u}_g \vec{u}_g\big) = -\alpha_g \nabla p + \nabla \cdot \overline{\tau}_g + \alpha_g \rho_g \vec{g} + K_{gs}(\vec{u}_g - \vec{u}_s) \tag{3}$$

$$\frac{\partial}{\partial t}(\alpha_s\rho_s\vec{u}_s)+\nabla\cdot(\alpha_s\rho_s\vec{u}_s\vec{u}_s)=-\alpha_s\nabla p-\nabla p_s+\nabla\cdot\overline{\tau}_s+\alpha_s\rho_s\vec{g}+K_{gs}(\vec{u}_g-\vec{u}_s) \tag{4}$$

where K_{gs} represents the interphase momentum transfer was described by the Gidaspow drag law (Eq. (5-7)), while particle–wall interactions were represented by the Johnson–Jackson boundary condition (Eq. (8-9)) with a restitution coefficient of 0.9 and specularity coefficient of 0.2.

Gidaspow drag model [6]:

For $\varepsilon_g > 0.8$:

$$K_{gs}=\frac{3}{4}C_D\frac{\varepsilon_s\varepsilon_g\rho_g\left|\vec{u}_s-\vec{u}_g\right|}{d_p}\varepsilon_g^{-2.65} \tag{5}$$

For $\varepsilon_g \leq 0.8$:

$$K_{gs}=150\frac{\varepsilon_s^2\mu_g}{\varepsilon_g d_s^2}+1.75\frac{\varepsilon_s\rho_g\left|\vec{u}_s-\vec{u}_g\right|}{d_s} \tag{6}$$

$$C_D=\frac{24}{\varepsilon_g Re_s}\left(1+0.15(\varepsilon_g Re_s)^{0.687}\right) \tag{7}$$

Johnson and Jackson partial slip model [8]:

$$\mu_s\frac{\partial U_s}{\partial x}=\frac{\pi\phi_s\rho_s\alpha_s g_0\sqrt{\theta_s}}{2\sqrt{3}\alpha_s^{\max}}U_s, \tag{8}$$

$$\kappa_s\frac{\partial\theta_s}{\partial x}=-\frac{\pi\phi_s U_s^2\rho_s\alpha_s g_0\sqrt{\theta_s}}{2\sqrt{3}\alpha_s^{\max}}-\frac{\pi\sqrt{3}\,\phi_s\rho_s\alpha_s g_0\left(1-e_w^2\right)\sqrt{\theta_s}}{4\alpha_s^{\max}}\theta_s. \tag{9}$$

where μ_s and κ_s are the viscosity and conductivity of the solid phase, and ϕ_s and e_w are the specularity coefficient and the particle–wall coefficient of restitution, respectively.

Numerical simulations were performed on a structured mesh of 56 × 200 cells (11,200 total), corresponding to a uniform grid spacing of 5 mm. Time integration employed an implicit first-order scheme with a base time step of $\delta t = 10^{-3}$s. Postprocessing was conducted in ParaView and with OpenFOAM utilities. Pressure drop and BER were computed and compared against both experimental data and results from recent CFD studies. To improve reliability, a new mass-conservation-based approach for BER calculation was tested alongside conventional midline methods.

3 Literature Review

The benchmark of Taghipour et al. [1] remains a cornerstone for validating CFD models of gas–solid fluidized beds, providing reliable experimental data on pressure drop and BER. Their work demonstrated the applicability of Eulerian–Eulerian two-fluid models (TFM) with drag laws such as Wen–Yu [4], Syamlal–O'Brien [5], and Gidaspow [6]. Because of its reproducibility, this case has been widely adopted in later validation studies. Several authors have extended the benchmark using different CFD solvers. Herzog et al. [9] compared Fluent, MFIX, and OpenFOAM, finding that global parameters like pressure drop were captured reasonably well. Londono [10] reported larger deviations, while Kusriantoko et al. [11] emphasized sensitivity to mesh resolution, boundary conditions, and particle restitution. Fatti [7] focused on OpenFOAM and showed that pressure drop was stable across numerical setups, but BER was strongly influenced by wall models. Liu [12] highlighted the role of adaptive time-stepping and averaging windows, which improved stability of results. A major source of disagreement across studies is the calculation of the BER. Herzog [9] and Liu [12] used pressure-drop methods, whereas Kusriantoko [11] applied a midline solid fraction approach, which tended to overpredict expansion. This inconsistency underlines the need for more reliable postprocessing techniques. More recent works expanded validation to different solvers and scales. Shi et al. [13] compared 2D and 3D models in Fluent, recommending 3D for accuracy but retaining 2D for sensitivity analysis. Reyes-Urrutia et al. [14] compared OpenFOAM and MFIX for fluidized beds with heat transfer, finding both reliable but MFIX slightly more accurate. Patil [15] and Armstrong [16] validated similar cases using CFX and Fluent, further confirming the robustness of Taghipour's benchmark [1] across platforms. In summary, the literature demonstrates that predictive accuracy depends strongly on drag models, boundary conditions, and numerical settings. Earlier OpenFOAM studies (Herzog [9], Londono [10]) reported significant deviations, whereas more recent works (Fatti [7], Kusriantoko [11]) suggest that careful parameter selection can yield accurate results. However, no study has yet applied the latest OpenFOAM v12 to this benchmark. The present work addresses this gap by assessing solver performance and proposing a new mass-conservation-based BER calculation method as an alternative to existing approaches.

4 Results and Discussion

The inherently transient processes of bubble coalescence and breakup generate significant pressure-drop oscillations within the fluidized bed [9]. To avoid the influence of these initial fluctuations, the pressure drop used for comparison was calculated as a time-averaged quantity only after the flow had reached a statistically steady state. Consistent with the procedure reported by Taghipour [1], the averaging of global parameters commenced after 3 s of simulated time.

The pressure drop results obtained by various researchers using OpenFOAM for Taghipour's setup are summarized in Tab. 1. All simulations considered here correspond to an inflow gas velocity of 0.38 m/s. Earlier numerical studies using OpenFOAM have reported pressure-drop values between 5027 Pa and 8064 Pa, compared with Taghipour's experimental measurement of 5423.398 Pa. The simulations by Herzog [9] and Londono [10] show the greatest departure from the experiment, predicting 7072 Pa and 8064 Pa, respectively. Conversely, results on the lower end of the reported range tend to more closely match the experimental value. Although Herzog (2012), Londono (2012), Fatti (2021), and Kusriantoko (2024) all used OpenFOAM, their reported values differ due to changes in solver versions and the availability of specific models. A noticeable trend appears: as the simulations become more recent, their deviation from the experimental benchmark decreases. The most up-to-date results, produced by Fatti and Kusriantoko, are in close agreement; however, because Fatti provides more extensive methodological details, their findings are adopted as the primary

Table 1: Pressure drop results obtained by various authors

Sources	Pressure [Pa]
Experiment	5423.398
Herzog	7072.423
Londono	8064.067
Fatti J&J	5067.370
Kusriantoko	5072.617

Table 2: Comparison

Time discretization	This work's dP [Pa]	Deviation from results of
10^{-5}s euler	5145.96	Fatti et al. is 1.5%
10^{-3}s euler	5132.88	Kusriantoko et al. is 1.2%

point of comparison in this work.

To assess the reliability of the present simulations, the parameter set used by Fatti was replicated. This resulted in a predicted pressure drop of 5145.96 Pa, differing from Fatti's value by only 1.5%. Likewise, reproducing the conditions reported by Kusriantoko produced a pressure drop of 5132.88 Pa, corresponding to a 1.2% deviation (Tab. 2). These results demonstrate that the solver version employed in this study (v12) yields predictions that closely align with the recent OpenFOAM investigations of Fatti and Kusriantoko et al., and that the discrepancies are considerably smaller than those observed in earlier works, such as those by Herzog and Londono.

According to the Ergun equation, which is applied in the Gidaspow drag model for $\varepsilon \leq 0.8$ (Eq. (6)), the pressure drop across a bed increases with increasing superficial gas velocity [17]. When the gas velocity reaches a value at which the drag force on the particles balances their weight ($m \times g$), the bed becomes fluidized. This velocity is referred to as the minimum fluidization velocity u_{mf}. Figure 1 presents the dependence of bed pressure drop on inflow air velocity reported by various authors. The results of the current simulations, as well as those of Kusriantoko and Herzog, are consistent with the prediction according to the Ergun equation, showing an increase in pressure drop with increasing velocity until the inflow velocity reaches u_{mf}. In contrast, the plots reported by Fatti and Londono exhibit deviations from this trend at low velocities. However, it is noticeable that Kusriantoko's pressure drop begins to level off later than both the current simulation results and Fatti's results. This discrepancy may be due to a lack of velocity points: there are insufficient data near u_{mf}, making it difficult to determine whether the pressure drop increases exactly up to u_{mf}. For velocities above $u_{mf} = 0.62$ m/s, the results of the current work closely match those of Fatti et al. and Kusriantoko et al.

Tab. 3 presents BER reported by different authors alongside the experimental value of 1.491 obtained by Taghipour. The results show significant variation across studies, with reported values ranging from 1.343 (Taghipour) to 1.721 (Kusriantoko's OpenFOAM).

It is clear that pressure drop is the difference of time-averaged and spatial-averaged (along boundary) pressure between inlet and outlet boundaries. However BER can be determined in three ways: from the pressure drop along a vertical midline of the bed (*midline* ΔP method);

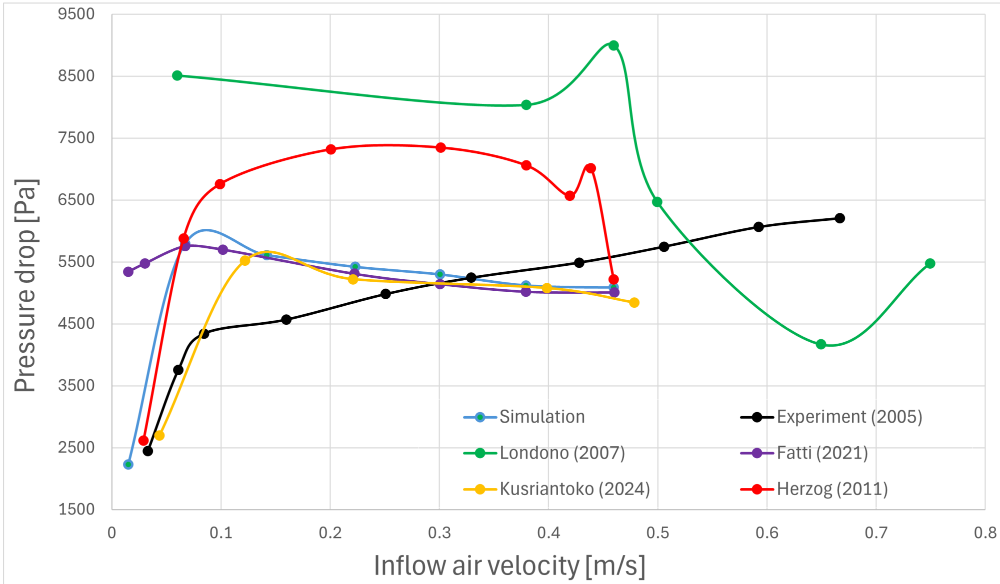

Figure 1: Bed pressure drop – inflow velocity relation comparison with the results of other researchers

Table 3: Bed expansion ratio BER results obtained by various authors

Sources	BER
Experiment	1.491
Herzog	1.538
Londono	1.380
Fatti J&J	1.554
Kusriantoko	1.721

from time-averaged gas fraction along midline (*midline gas fr.* method); from calculating sum of particles time-averaged mass (*domain mass* method).

Herzog [9] and Liu [12] applied the *midline* ΔP but did not specify the criterion referred to as the *threshold*, calculated by Eq. (10). The bed ratio is defined as the ratio between the bed height at which the threshold reaches 1 and the initial bed height:

$$\text{threshold} = \frac{P_{\text{inlet}} - P(h)}{P_{\text{inlet}} - P_{\text{outlet}}}. \tag{10}$$

Regarding the *midline gas fr.* method, Kusriantoko [11] proposed a criterion based on the gas fraction $\varepsilon_g(h)$ along the midline. In this method, the height at which the *threshold* that is gas fraction equals 0.99 defines the extent of BER, and the BER is computed as the ratio between this height and the initial bed height.

The *domain mass* method was developed in the present work. It is based on integrating the solid mass (or, equivalently for incompressible flow, the solid volume fraction) over all cells of the computational domain. The bed ratio is defined as the height below which the total solid mass in the bed is nearly equal to the total solid mass in the entire domain. The corresponding *threshold* is computed using Eq. (11):

$$\text{threshold} = \frac{\sum_{j=1}^{j(h)} \sum_{i=1}^{56} \varepsilon_s}{\varepsilon_{s,\text{init}} H_0}, \tag{11}$$

where i and j denote the horizontal and vertical cell indices, respectively.

Kusriantoko used the $\delta t = 10^{-3}$s euler scheme for time discretization. He determined BER at height where the air fraction reached 0.99 and got $H/H_0 = 1.721$. The value of the current work by the scheme and method with a threshold such as Kusriantoko is equal to 1.658. That's deviation from Kusriantoko's is 3.6%. However, the values taken by Kusriantoko overcome the values of the results of all other authors. Moreover, the current work value by Kusriantoko's method overcomes the current work values by other methods too according to Tab. 4. For this reason, the Kusriantoko's method is discarded for the following studies.

Table 4: Bed expansion ratio BER calculated by time discretization of $\delta t = 10^{-3}$s, scheme euler

Threshold/Method	midline ΔP	midline gas fr.	domain mass
0.950	1.382	1.621	1.438
0.980	1.482	1.646	1.525
0.990	1.533	1.658	1.562

Fatti used $\delta t = 10^{-5}$s euler and got $H/H_0 = 1.554$. Among the results by different methods for H/H_0 shown in Tab. 5, results taken by *midline* ΔP method with 0.99 threshold and by *domain mass* method with 0.98 threshold are closest to Fatti's result. It approves the use of *midline* ΔP method with 0.99 threshold or domains mass method with 0.98 threshold.

Table 5: Bed expansion ratio BER calculated by time discretization of $\delta t = 10^{-5}$s, scheme euler

Threshold/Method	midline ΔP	domain mass
0.950	1.382	1.462
0.980	1.482	1.537
0.990	1.533	1.587

Fig. 2 shows BER dependence on inflow air velocity of different authors. Across all cases, the BER increases with inflow velocity, and the growth patterns similar. Kusriantoko's OpenFOAM results show the highest expansion values overall, due to the use of method *midline gas fr.*. Plot of Fatti and this works simulation almost overlaps each other, meaning that they validate each other. This simulations plot was done by method *domain mass* (plots of by method *midline* ΔP and by method *domain mass* gave the same plot). Both of these plots are close to experimental result. Herzog reported not as smooth as Kusriantoko, but closely matching experimental and simulation trends. In contrast, Londono [10] observed comparable behavior at low velocities but significant fluctuations above 0.4 m/s, with expansion ratios oscillating between 1.45 and 1.8. These deviations suggest flow instabilities not present in Herzog's data.

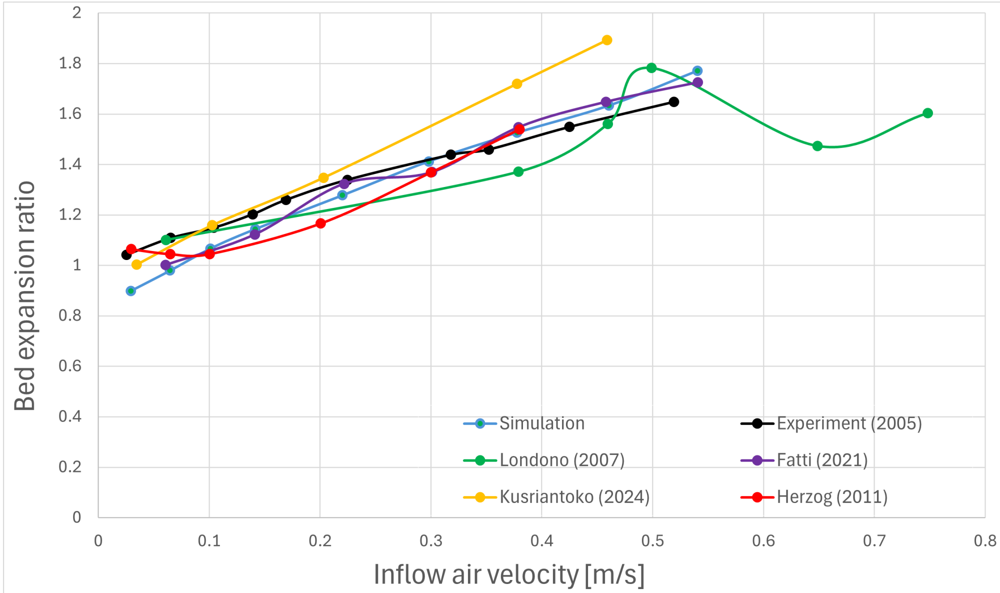

Figure 2: Bed expansion ratio BER – inflow velocity relation comparison with the results of other researchers

5 Conclusion

This work assessed the capability of OpenFOAM v12 to simulate gas–solid fluidized bed hydrodynamics against the benchmark of Taghipour et al. Results confirmed that the solver provides accurate pressure drop predictions, with deviations under 6% relative to experiments and under 2% compared with recent CFD studies. The newly proposed mass-conservation-based method for calculating BER was shown to deliver more consistent and experimentally aligned results than conventional midline solid fraction approaches, which tend to overpredict expansion. Overall, the study demonstrates that OpenFOAM v12 can reliably reproduce key hydrodynamic indicators of fluidized beds while offering improved postprocessing strategies. These advances support the application of CFD in industrial design, scale-up, and optimization of fluidized bed reactors.

6 Acknowledgements

The authors express their sincere gratitude to the “Heat Management” research team at the LEMTA R&D Center of the University of Lorraine and CNRS (Nancy, France) for their valuable support. The authors also acknowledge the organizers of the double-degree Master’s program between Al-Farabi Kazakh National University (Kazakhstan) and the University of Lorraine (France), specializing in “7M05405 — Mechanics and Energy,” for their contribution to the academic preparation of Master’s graduates. The authors are particularly grateful to Professor Olivier Botella for supervising Master’s student Nazerke Momysh in her CFD modeling work on high-temperature thermal storage in a fluidized-bed reactor.

References

[1] Taghipour, F., Ellis, N., Wong, C.: Experimental and computational study of gas–solid fluidized bed hydrodynamics. Chemical Engineering Science 60, 6857–6867 (2005).

[2] Khawaja, H.: Review of the phenomenon of fluidization and its numerical modelling techniques. The International Journal of Multiphysics 9(4), 397– (2015). doi:10.1260/1750-9548.9.4.397.

[3] Abdidin, A., Kereikulova, A., Toleukhanov, A., Botella, O., Kheiri, A., Belyayev, Y.: Two-dimensional CFD analysis of a hot water storage tank with immersed obstacles. Journal of Mechanics, Mathematics and Computer Science (JMMCS) 3(123), 58–74 (2024). doi:10.26577/JMMCS2024-v123-i3-7.

[4] Wen, C.Y., Yu, Y.H.: Mechanics of fluidization. Chemical Engineering Progress Symposium Series 62, 100–111 (1966).

[5] Syamlal, M., O'Brien, T.J.: The derivation of a drag coefficient formula from velocity–voidage correlations. Technical Note, U.S. Department of Energy, Office of Fossil Energy, NETL, Morgantown, WV (1987).

[6] Gidaspow, D.: Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions. Academic Press, (1994).

[7] Fatti, V., Fois, L.: CFD modeling of gas-solid fluidized beds in OpenFOAM: a comparison between the Eulerian-Eulerian and Eulerian-Lagrangian methods. M.S. thesis, Politecnico di Milano, School of Industrial and Information Engineering (2021).

[8] Johnson, P.C., Jackson, R.: Frictional–collisional constitutive relations for granular materials, with application to plane shearing. Journal of Fluid Mechanics 176, 67–93 (1987).

[9] Herzog, N., Schreiber, M., Egbers, C., Krautz, H.J.: A comparative study of different CFD-codes for numerical simulation of gas–solid fluidized bed hydrodynamics. Computers and Chemical Engineering (2011).

[10] Londono, A., Londono, C., Molina, A., Chejne, F.: Simulation of gas–solid fluidized bed hydrodynamics using OpenFOAM. In: OpenFOAM International Conference (2007).

[11] Kusriantoko, P., Daun, P.F., Einarsrud, K.E.: A comparative study of different CFD codes for fluidized beds. Dynamics 4, 475–498 (2024). doi:10.3390/dynamics4020025.

[12] Liu, Y. et al.: A critical comparison of the implementation of granular pressure gradient term in Euler–Euler simulation of gas–solid flows. Computers and Fluids (2010).

[13] Shi, H., Komrakova, A., Nikrityuk, P.: Fluidized beds modeling: Validation of 2D and 3D simulations against experiments. Powder Technology 343, 479–494 (2019).

[14] Reyes-Urrutia, A., Venier, C., Mariani, N.J., Nigro, N., Rodriguez, R., Mazza, G.: A CFD Comparative Study of Bubbling Fluidized Bed Behavior with Thermal Effects Using the Open-Source Platforms MFiX and OpenFOAM. Fluids 7(1) (2022).

[15] Patil, D.J., Smit, J., van Sint Annaland, M., Kuipers, J.A.M.: Wall-to-Bed Heat Transfer in Gas–Solid Bubbling Fluidized Beds. AIChE Journal 52(1) (2006). doi:10.1002/aic.10590.

[16] Armstrong, L.M., Gu, S., Luo, K.H.: Study of wall-to-bed heat transfer in a bubbling fluidised bed using the kinetic theory of granular flow. International Journal of Heat and Mass Transfer 53(21–22), 4949–4959 (2010). doi:10.1016/j.ijheatmasstransfer.2010.06.025.

[17] Cocco, R.F. et al.: Particle Clusters in and above Fluidized Beds. Powder Technology 203, 3–11 (2010).

[18] Loha, C., Chattopadhyay, H., Chatterjee, P.K.: Euler–Euler CFD modeling of fluidized bed: Influence of specularity coefficient on hydrodynamic behavior. Particuology 11(6), 673–680 (2013). doi:10.1016/j.partic.2012.08.009.

[19] Li, T., Grace, J., Bi, X.: Study of wall boundary condition in numerical simulations of bubbling fluidized beds. Powder Technology (2010).

Авторлар туралы мәлімет

Момыш Назерке Бақытжанқызы – әл-Фараби атындағы Қазақ ұлттық университеті (Қазақстан) мен Лотарингия университеті (Франция) арасындағы қос дипломды бағдарламаның "7M05405 - Механика және энергетика"мамандығы бойынша магистратураны тәмамдады. Қазақстан Республикасы Ғылым және Жоғары Білім Министрлігі тарапынан берілетін Абай-Верн бір жылдық стипендиясының иегері (Алматы қаласы, Нанси қаласы, Қазақстан, Франция, nazerke.momysh7@etu.univ-lorraine.fr; nazerke.momysh@gmail.com);

Беляев Ержан Келесұлы (корреспондент автор) - PhD, қауымдастырылған профессор, әл-Фараби атындағы Қазақ ұлттық университетінің Механика кафедрасының профессор-зерттеушісі. әл-Фараби атындағы Қазақ ұлттық университеті мен Лотарингия университеті арасындағы "7M05405 - Механика және энергетика"мамандығы бойынша қос дипломды бағдарламаның Қазақ тарапынан координаторы (Алматы қаласы, Қазақстан, yerzhan.belyaev@kaznu.edu.kz; yerzhan.belyayev@gmail.com);

Оливье Ботелла - PhD, Лотарингия университетінің қауымдастырылған профессоры, Лотарингия университеті жанындағы "Жылу менеджменті"зертханасының LEMTA атты энергетика, теориялық және қолданбалы механика зерттеу орталығының ғылыми қызметкері. әл-Фараби атындағы Қазақ ұлттық университеті мен Лотарингия университеті арасындағы "7M05405 - Механика және энергетика"мамандығы бойынша қос дипломды бағдарламаның Француз тарапынан координаторы (Нанси қаласы, Франция, olivier.botella@univlorraine.fr);

Информация об авторах

Назерке Момыш – выпускница двойной магистерской программы между Казахским национальным университетом им. Аль-Фараби (Казахстан) и Лотарингским университетом (Франция) по специальности «7M05405 – Механика и энергия». Получательница годичной стипендии Абай-Верн от Министерства науки и высшего образования Республики Казахстан (Алматы, Нанси, Казахстан, Франция, nazerke.momysh7@etu.univ-lorraine.fr; nazerke.momysh@gmail.com);

Ержан Беляев (автор для корреспонденции) – кандидат наук, доцент, профессор-исследователь кафедры механики Казахского национального университета им. Аль-Фараби. Координатор программы двойного диплома (с казахстанской стороны) между Казахским национальным университетом им. Аль-Фараби и Лотарингским университетом по специальности «7M05405 – Механика и энергия» (Алматы, Казахстан, yerzhan.belyaev@kaznu.edu.kz; yerzhan.belyayev@gmail.com).

Оливье Ботелла – кандидат наук, доцент Лотарингского университета, Франция. Научный сотрудник лаборатории «Управление теплом» Научно-исследовательского центра энергетики, теоретической и прикладной механики (LEMTA), Лотарингский университет, Франция. Координатор программы двойного диплома (с французской стороны) между Казахским национальным университетом им. Аль-Фараби и Лотарингским университетом по специальности «7M05405 - Механика и энергия» (Нанси, Франция, olivier.botella@univ-lorraine.fr);

Information about authors

Nazerke Momysh – graduated the dual-degree Master program between Al-Farabi Kazakh National University (Kazakhstan) and the University of Lorraine (France) in the specialty "7M05405 - Mechanics and Energy". Recipient of the one-year Abai-Verne scholarship from the Ministry of Science and Higher Education of the Republic of Kazakhstan (Almaty, Nancy, Kazakhstan, France, nazerke.momysh7@etu.univ-lorraine.fr; nazerke.momysh@gmail.com);

Yerzhan Belyayev (corresponding author) –PhD, Associate Professor, Professor-Researcher of the Department of Mechanics at Al-Farabi Kazakh National University. Coordinator of the dual-degree program (from the Kazakh side) between Al-Farabi Kazakh National University and the University of Lorraine in the specialty "7M05405 - Mechanics and Energy"(Almaty, Kazakhstan, yerzhan.belyaev@kaznu.edu.kz; yerzhan.belyayev@gmail.com).

Olivier Botella – PhD, Associate Professor at the University of Lorraine, France. Researcher at the "Heat Management"Lab of the Energy, Theoretical and Applied Mechanics Research Center (LEMTA), University of Lorraine, France. Coordinator of the dual-degree program (from the French side) between Al-Farabi Kazakh National University and the University of Lorraine in the specialty "7M05405 - Mechanics and Energy"(Nancy, France, olivier.botella@univ-lorraine.fr);

Received: November 29, 2025
Accepted: December 17, 2025