OpenFOAM NUMERICAL SIMULATION AND EXPERIMENTAL VALIDATION OF HIGH CAPACTY LONG DISTANCE FLY ASH PNEUMATIC CONVEYING FROM A 620 MW THERMAL POWER PLANT

Nikola KARLIČIĆ [*], *Darko RADENKOVIĆ, Milan RAKOVIĆ, Marko OBRADOVIĆ, Dušan TODOROVIĆ*

University of Belgrade, Faculty of Mechanical Engineering, Kraljice Marije 16, 11120 Belgrade 35, Serbia

[*] Corresponding author; E-mail: nkarlicic@mas.bg.ac.rs

The authors conducted pressure drop numerical simulations of multiphase gas-solid flow characteristics, within a fly ash pneumatic conveying system, in 620 MW thermal power plant. The objective of this study is to verify the OpenFOAM model by comparing the numerical results with pressure measurements taken along the considered pipeline. The numerical model is developed on the base of extensive experimental research of high-capacity long-distance pneumatic conveying system for Kolubara lignite fly ash. The numerical simulations of pneumatic conveying were performed using the Euler-Euler approach in OpenFOAM, that is twoPhaseEulerFoam solver, for the air mass flow rate of 5500 Nm^3/h, the fly ash mass flow rate 77 t/h. The calculation input air pressure at the pipeline outlet was assumed to be as measured approximately 234 kPa, the air-ash mixture temperature was 373,15K, the mean ash particle diameter of 0.128 µm, and physical density of ash 2100 kg/m^3. Numerical mesh is generated as an O-grid for the first two pipeline sections, first with a length of 90 m, a diameter of 0.2604 m, and an inclination of 1.885 degree, and the second with a length of 102 m, a diameter of 0.3097 m, and an inclination of 2.163 degree, while every mesh cell is substantially larger than the diameter of the ash particle. The applied Euler-Euler approach enables a comprehensive investigation of the complex dynamics involved in pneumatic conveying, considering the two-phase nature of the system and providing valuable insights into the particle behavior, pressure drop, and other key parameters. The comparison of the experimental and numerical model simulations data show good agreement regarding pressure drop. Although Euler-Euler model is complex in terms of necessary closure models, it might prove itself as reliable for simulated conditions in future studies.

Key words*: fly ash, pneumatic conveying, numerical simulations, Euler-Euler approach, OpenFOAM, twoPhaseEulerFoam solver;*

1. Introduction

Coal-fired power plays tremendously important role ensuring almost 40% of global electricity [1]. By far the most common fuel used to produce electricity in the Western Balkans is lignite, which is accounted for 55% of the region's total gross power output in 2021 (44.931 GWh) [2]. The primary fuel used in Serbian thermal power plants (TPPs) is lignite, with a share of 62% in electricity production in 2021 [2].

Fly ash stands out as the primary byproduct from the combustion of pulverized coal in TPP. Pneumatic conveying systems are preferred solution for swiftly evacuating substantial amounts of fly ash over extensive distances. Designing these systems poses challenges, operational difficulties in daily use are common, and frequent fly ash transport delays may result in TPP shutdowns [3-6]. The efficiency of fly ash pneumatic conveying system hinges entirely on the its characteristics, and there are extreme variations even within the same batch [7]. Comprehensive understanding of the material properties for pneumatic transport is crucial to avoid operational problems [3, 5, 8]. On the other hand, simple data on basic material properties will not be strong enough for determining minimum transport speeds, pressure drops, etc., particularly when pneumatic transport is being used over long distances and in large-diameter pipes.

The computational approach to pneumatic conveying provides constant progress in this particular field, it enables optimization between investment and operational costs. However, a deeper understanding of the fundamental physical characteristics of the process is required for practical use of this advancement [8]. Existing models and correlations reveal difficulties in application due to poorly defined conditions, constraints, or complex parameters that are challenging or impossible to determine. Gas-solid multiphase flow modeling is rather complex, and the nature of fly ash provides additional issues regarding this task. The variability in fly ash properties within the same plant adds to the complexity of the modeling process [8]. Considering that experimental work can serve as a crucial reality check for computational models, providing model validation with real experimental data is of the highest importance [9]. In addition to experimental measurements, CFD modeling may be a useful tool to acquire an extensive understanding of the fly ash pneumatic conveying.

Besides CFD, the studies [3, 4, 10] conducted extensive experimental research and theoretical analysis on fly ash long-distance pneumatic conveying issues. Numerical simulations of pneumatic conveying in pipes (up to 10 m length) using a two-phase model are in good agreement with experimental data [11-14]. However, numerical studies of the pneumatic transport of solids through long inclined pipes using a two-phase model are scarce.

The main goal of this study is to conduct numerical simulations of the pneumatic conveying of lignite fly ash through stepped diameter, inclined, high-capacity long duct, based on real operating conditions in a 620 MW TPP unit. Euler-Euler approach was applied to establish numerical model in OpenFOAM, and twoPhaseEulerFoam solver was used to conduct simulations. The model validation involved comparing the simulations results with experimental data of pressure measurements along the pipeline.

2. Challenges in modelling of pneumatic conveying of fly ash

Since the early 20th century, pneumatic conveying of bulk solid materials has been utilized on a large scale in industry [15]. For more than 100 years pneumatic conveying has attracted significant attention amongst engineers and others interested in cost-effective, yet simple handling of bulk solid materials [16], due to numerous advantages over conventional methods. Nonetheless, the use of pneumatic conveying in a vast variety of industrial applications is still affected by numerous problems [17, 18].

Fly ash from TPP along with transport air generates a two-phase mixture, and gas-solid multiphase flow modelling is very challenging, while the nature of fly ash leads to even more problems for the modelling work. Analyzing gas-solid flows requires perception of the behavior of particles in a gas medium, identifying the regimes of the flow and appropriate models, and at the end selecting and applying adequate models for each regime [4, 15, 19-21]. There are two basic modes of pneumatic conveying: dilute (lean) phase and dense phase [22]. Nevertheless, there is still questionable and uncertain boundary between these two phases, and demanding issue for an engineer to decide between the dense and dilute phase. Besides dilute and dense regimes, one of the common is also a fluidized regime. The inability to precisely understand particle behavior in each regime, and how material properties affect the conveying flow regime within the pipeline is one of the biggest challenges to accurately predict the performance of pneumatic conveying systems [22]. The very same reason makes extremely challenging to apply existing models as a consequence of imperfectly defined conditions, limitations and presence of complex parameters demanding to determine.

Computational fluid dynamics (CFD) modeling has become a powerful tool in past few decades for delivering both qualitative and quantitative insight into complex gas–solid flows, owing primarily to rapid advancements in computer hardware and numerical algorithms [23]. The paper [15] discusses a wide range of aspects and numerous CFD studies that have been conducted along with different model approaches for various geometries and solid materials. Flow regimes such as dispersed, stratified, dune flow, slug and plug flows, with wide range of solid loadings, horizontal pipe flows, vertical upward flows and flows in horizontal-to-vertical bends/elbows, particle diameters in range from micrometers to millimeters, and particle density 500-4000 kg/m^3.

There are a several types of CFD models for predicting gas-solid flow, and mainly, two approaches are used, the Eulerian–Eulerian method for granular flows and the Eulerian–Lagrangian method [24].

The Euler-Lagrangian method includes Lagrangian discrete phase model, dense discrete phase model incorporated with the kinetic theory of granular flow, CFD-discrete element method and computational particle fluid dynamics numerical scheme incorporated with the multiphase-particle-in-cell. Additionally, enhanced conventional models, among them ANSYS Fluent's dense discrete phase model incorporates the discrete element technique. Considering particle-particle interactions and the numerical techniques for solving the equations vary depending on the model for both approaches [15]. Each model has advantages as well as limitations of its own, thus the most appropriate model should be used based on the main factors to consider.

Euler-Euler approach typically requires less computational resources than Euler-Lagrangian methods [23], yet having significant limitations of this approach is considering variations in particle properties, such as wide particle size distributions, density diversification, and sphericity considerations. At the same time, despite greater demands in computational resources, the Eulerian-Lagrangian approach may consider flows with a range of particle sizes, shapes, and velocities [24].

The most important challenge in modelling fly ash pneumatic transport is random changing of the properties of the fly ash. This includes a very wide range of the fly ash particle size distribution, non-spherical in shape of the fly ash particle, the size and shape of the fly ash particles can be changed while conveying due to degradability, and the most important fly ash properties are different from batch to batch [3-5, 10], even within the single batch of the same material. Modelling a wide particle size distribution is a major challenge due to different timescales associated with different sizes. Dividing the full-size distribution to different particle distribution classes and carrying out simulations for narrow ranges of distributions would be an initial suggestion. In Eulerian-Lagrangian approaches, the mesh size needs to be larger than the particle size [25-27]. At the same time, the size of the computational cells should be small enough to capture the fluid pattern accurately [15]. Studies related to non-spherical particles (ellipsoids, discs, cylinders and other irregular shapes) address the effect of non-sphericity on drag force and relevant modifications for drag coefficients, importance of considering wall-roughness effect and rotational motion of non-spherical particles compared to spherical particles and applicability of the multi-sphere discrete element method [15, 25, 28-31]. Previous attrition and abrasion studies can be useful for modeling degradation of particles during pneumatic conveying [15]. CFD approach was used to determine stress conditions to predict attrition level [32, 33], or comminution functions were implemented into in CFD-DEM model [34, 35]. Another challenge would be a lack of full-size rig or large-scale industry system numerical simulations. This is understandable because simulating huge pneumatic conveying, instead of focusing only on a section of the whole system, is computationally demanding, particularly with Eulerian-Lagrangian approach. Defining boundary conditions, especially at the inlets of certain simulated sections is a huge challenge, and experimental setups should be well-defined and verified, if the results are to be compared [15].

CFD approaches to gas/solid flow systems and fluidization are developing into sophisticated instruments for process design of these kinds of systems. The current models can forecast the following phenomena: bubble formation and flow behavior in a bubbling fluidized bed; core annular flow; cluster formation; and the descent of the clusters at the wall of the riser section of the circulating fluidized bed [36]. There are still numerous research possibilities in investigating of reducing the constraints of the CFD in fly ash pneumatic conveying simulations, such as optimizing the governing and constitutive equations to include multiple particle, surface and phase interactions. To make the codes useful, computing time must be reduced while solving grid independent and stable three-dimensional equations for complex geometries. The accurate predictions using CFD codes fully depend on detailed information on an exact description of inlet and outlet geometries, proper measurement techniques, data on the boundary of the gas/solid flow system, such as particle collision with the boundary, and particle slip velocity and fluctuation for the different flow regimes [36].

Despite advancements in computational capability and advanced tools, the complexity of gas/particle flow systems, realistic simulations and accurate experimental data are crucial for improving efficiency in gas/particle flow systems [36]. Increasingly, more attention has been directed to combined experimental and numerical investigation of multiphase flow characteristics, where simulation results turn to be reasonable, with promising performance within a broad solids loading ratio, but further work is necessary to make it more reliable [37].

Development of a universal model considering all essential and complex physics involved in gas-solids flows aspects is challenging and a step-wise approach [15]. Achievement of this understanding involves the development of experimental measurement techniques, experimentally verified multiphase flow equations and numerical simulation tools [36].

3. Numerical model

To simulate pneumatic transport of solids in pipes, two large groups of numerical models are available, discrete element model (DEM) and two-phase model. DEM studies [38, 39] are usually conducted for relatively smaller pipe lengths and a reasonable number of particles.

In this model, particles are represented as discrete objects. The soft-sphere model allows the calculation of inter-particle collisions [40]. However, in this study, because of the long duct length and large number of particles, it is computationally expensive to conduct numerical simulations using DEM. Therefore, it is decided to investigate the pneumatic conveying of solids in long ducts using a two-phase model.

Numerical simulations are conducted and analyzed for such a flow configuration using the OpenFOAM two-fluid model. The numerical simulations input data are based on experimental data obtained from studies [3, 4]. The results are obtained for the air mass flow rate 5500 Nm^3/h and the ash mass flow rate 77 t/h. The calculation input air pressure at the pipeline outlet is assumed to be as measured approximately 234 kPa, the air-ash mixture temperature was 373,15 K, the mean ash particle diameter 128 μm and physical density of ash of 2100 kg/m^3.

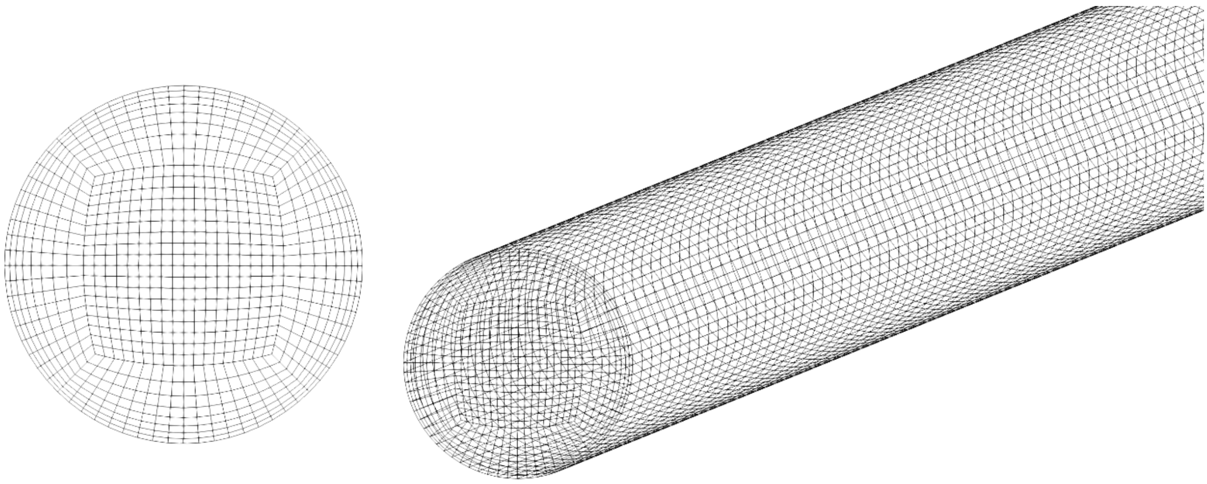

Figure 1. Sketch of a typical O-grid mesh used in numerical simulations

Numerical mesh (Fig. 1) is generated as an O-grid for the first two sections of inclined telescopic pipeline 1 from studies [3, 4]. First section has length of 90 m, diameter of 0.2604 m, and an inclination of 1.885 degree, the second has length of 102 m, diameter of 0.3097 m, and an inclination of 2.163 degree, while every mesh cell is substantially larger than the diameter of the ash particle. Number of mesh cells is 20000 in first section and 33120 in second section.

The numerical simulations of pneumatic conveying were performed using the Euler-Euler approach in OpenFOAM, that is twoPhaseEulerFoam solver.

3.1. Governing equations

To conduct numerical simulations of ash pneumatic conveying, the Euler-Euler two-phase model has been used. In this approach, both gas and solid phases are represented as continuums, and the corresponding equations are solved in the Eulerian frame. In numerical simulations, solver twoPhaseEulerFoam has been used, from OpenFOAM v1706.

The continuity and momentum equations for the gas phase are, respectively [41]:

$$\partial_{\mathrm{t}}\left(\varepsilon_g \rho_g\right)+\nabla \cdot\left(\varepsilon_g \rho_g \mathrm{u}_g\right)=0 \tag{1}$$

$$\partial_{\mathrm{t}}\left(\varepsilon_g \rho_g \mathrm{u}_g\right)+\nabla \cdot\left(\varepsilon_g \rho_g \mathrm{u}_g \mathrm{u}_g\right)=-\nabla p_g-\nabla \cdot\left(\varepsilon_g \tau_{\mathrm{g}}\right)-\varepsilon_g \rho_g \mathrm{g}-\mathrm{f}_{\mathrm{d}}+\varepsilon_s \nabla p_g \tag{2}$$

while these equations for solid phase are:

$$\partial_{\mathrm{t}}\left(\varepsilon_s \rho_s\right)+\nabla \cdot\left(\varepsilon_s \rho_s \mathrm{u}_s\right)=0 \tag{3}$$

$$\partial_{\mathrm{t}}\left(\varepsilon_s \rho_s \mathrm{u}_s\right)+\nabla \cdot\left(\varepsilon_s \rho_s \mathrm{u}_s \mathrm{u}_s\right)=-\nabla p_s-\nabla \cdot\left(\varepsilon_s \tau_{\mathrm{s}}\right)-\varepsilon_s \rho_s \mathrm{g}-\mathrm{f}_{\mathrm{d}}+\varepsilon_s \nabla p_g \tag{4}$$

where ε_g, ρ_g, u_g, ug, τ_{g}, p_g are phase fraction, density, velocity, stress tensor and pressure of gas and similarly, corresponding quantities for solid phase are denoted with subscript ”s”. Momentum interaction between phases is taken into account through the force density term f_{d}. This term is defined as [41]:

$$\mathrm{f}_{\mathrm{d}}=\beta\left(\mathrm{u}_g-\mathrm{u}_s\right) \tag{5}$$

where β is drag coefficient. In this study, the drag coefficient is taken according to Gidaspow [42]:

Ergun [43] correlation is used if $\varepsilon_g<0.8$:

$$\beta=150 \frac{\varepsilon_s \mu_g}{\varepsilon_g d_p^2}+1.75 \frac{\rho_g \varepsilon_s}{d_p \varepsilon_g}\left|\mathrm{u}_g-\mathrm{u}_s\right| \tag{6}$$

and Wen and Yu [44] [8] correlation if $\varepsilon_g>0.8$:

$$\beta=\frac{3}{4} C_D \frac{\varepsilon_s \rho_g\left|\mathrm{u}_g-\mathrm{u}_s\right|}{d_p} \varepsilon_g^{-2.65} \tag{7}$$

where

$$C_D=\begin{cases}\dfrac{24}{\mathrm{Re}_{\mathrm{p}}}\left[1+0.15\left(\mathrm{Re}_{\mathrm{p}}\right)^{0.687}\right], & \mathrm{Re}_{\mathrm{p}}<1000 \\ 0.44 & \mathrm{Re}_{\mathrm{p}}>1000\end{cases} \tag{8}$$

and

$$\mathrm{Re_p} = \frac{\varepsilon_s \rho_g \left| \mathrm{u}_g - \mathrm{u}_s \right| d_p}{\mu_g} \tag{9}$$

Solid phase stress tensor has the general form as for the case of Newtonian fluid:

$$\tau_g = \mu_s \left(\nabla \mathrm{u}_s + \nabla^T \mathrm{u}_s \right) - \left(\lambda_s - \frac{2}{3}\mu_s \right) \left(\nabla \cdot \mathrm{u}_s \right) \mathrm{I} \tag{10}$$

In two-phase fluid model, the effect of inter-particle collisions is included through solid's pressure p_s and solid's shear and bulk viscosity μ_s and λ_s, respectively. According to the kinetic theory of granular flows, these quantities depend on the granular temperature, $\Theta = \langle \mathrm{C} \cdot \mathrm{C} \rangle$, where C represents particle fluctuation velocity.

The governing equation for granular temperature is [45, 46]:

$$\frac{3}{2}\left[\partial_t \left(\varepsilon_s \rho_s \Theta \right) + \nabla \cdot \left(\varepsilon_s \rho_s \Theta \mathrm{u}_s \right) \right] = \tau_s : \nabla \mathrm{u}_s - \nabla \cdot \mathrm{q} - D_1 - D_2 \tag{11}$$

The first term on RHS represents the production of granular temperature. The second term represents diffusion of granular temperature [47], where flux of fluctuating energy is:

$$\mathrm{q} = -k \nabla \Theta \tag{12}$$

and k is conductivity of granular temperature. In this work, this conductivity is [42]:

$$k = \rho_s d_p \sqrt{\Theta} \left(2\varepsilon_s^2 g_0 \cdot (1+e) / \sqrt{\pi} + \frac{9}{8.0}\sqrt{\pi} g_0 \cdot 0.5 (1+e) \varepsilon_s^2 + \frac{15}{16}\sqrt{\pi}\varepsilon_s + \frac{25}{64}\sqrt{\pi} / (1+e) g_0 \right) \tag{13}$$

where e is coefficient of restitution and radial distribution function is from Sinclair and Jackson [48]:

$$g_0 = \left(1 \Big/ 1 - \left(\frac{\varepsilon_s}{\varepsilon_{\max}} \right)^{1/3} \right) \tag{14}$$

Dissipation of energy due to inelastic interparticle collisions is [42]:

$$D_1 = 3\left(1 - e^2\right) \varepsilon_s^2 \rho_s g_0 \Theta \left(\frac{4}{d_p}\sqrt{\frac{\Theta}{\pi}} - \nabla \cdot \mathrm{u}_s \right) \tag{15}$$

The rate of energy dissipation D_2 results from the work of fluctuating force exerted by gas on particles [46]. This term is modeled as:

$$D_2 = -3\beta \cdot \Theta - \frac{\beta^2 d_p \left|\mathrm{u}_s - \mathrm{u}_g\right|^2}{4\varepsilon_s \sqrt{\pi \cdot \Theta}} \tag{16}$$

Solids phase pressure and solid phase bulk velocity are given as [47]:

$$p_s = \rho_s \varepsilon_s \Theta \left[1 + (1+e) g_0 \varepsilon_s\right] \tag{17}$$

$$\lambda_s = \frac{4}{3}\varepsilon^2 d_p g_0 (1+e)\Theta^2 \frac{1}{\sqrt{\pi}} \tag{18}$$

Solid phase shear velocity is calculated as in [42]:

$$\mu_s = d_p \sqrt{\Theta}\left(\frac{4}{5}\varepsilon_s^2 g_0 \cdot (1+e) / \frac{1}{\sqrt{\pi}} + \frac{1.0}{15.0}\sqrt{\pi} g_0 (1+e)\varepsilon_s^2 + \frac{1}{6}\sqrt{\pi}\varepsilon_s + \frac{10}{96}\sqrt{\pi}\frac{1}{(1.0+e) g_0}\right) \tag{19}$$

3.2. Boundary and initial conditions

To reduce computational costs, the flow is analyzed section by section. The computational domain is an inclined pipe with a constant diameter. Phase volume fractions and phase velocities are specified as fixed values at the bottom of the pipe inlet section. The Neumann boundary condition is used at the pipe outlet for all variables, except for pressure. Pressure is specified as a fixed value at the pipe outlet. The flow is considered isothermal, with a temperature of 373K. At the pipe wall, the no-slip boundary condition is used for both phases.

4. Experimental data and validation of numerical simulations

The numerical simulations validation is based on experimental data from studies [3, 4], conducted to quantify pressure changes along fly ash pneumatic conveying pipelines within considered thermal power plant, from the blow tanks to the silo. Pipelines are telescopic and inclined, with length of approximately 600 m, Fig. 2. The lengths of pipe sections are as follows 90 m, 102 m, 293 m and 40 m, diameters 0.2604 m, 0.3097 m, 0.3396 m and 0.3888 m, and the inclinations 1.885, 2.163, 6.073 and 1.469 degrees. This study used data for the first two sections of pipeline 1 given in [3, 4]. In addition to pressure measurements, studies [3, 4] also analyzed other operating parameters of the considered TPP. The electric power at the generator is 606.4-622.6 MW, the lignite consumption 756-786 t/h, air flow 5500 Nm^3/h, the actual mass flow of ash through pipeline is 77 t/h, density of fly ash 1800-2400 kg/m^3, the mean ash particle diameter 115-141 μm, and the average value of air pressure at the inlet cross section for pipeline 1 is 303 kPa [3, 4].

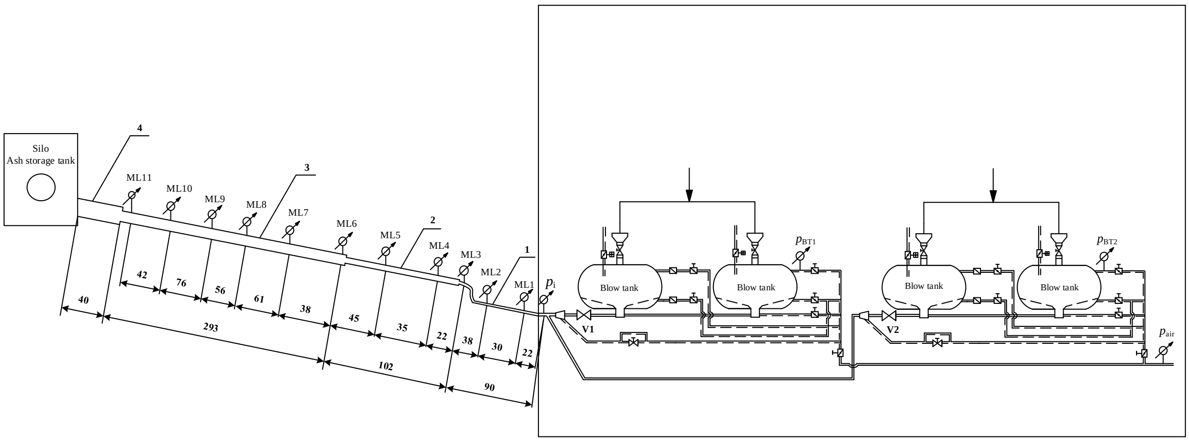

Figure 2. Lignite ash transport pipeline at the thermal power plant [3, 4]

The approach used to validate model was to determine the degree to which the numerical model predicted the pressure drop on approximately 200 m fly-ash pneumatic conveying through stepped-diameter telescopic pipeline from considered thermal power plant to the silo. The exact same dataset was used to successfully verify numerical simulations of fly ash long-distance pneumatic conveying in FORTRAN, based on the experimental conditions within the discussed thermal power plant [3, 4, 10]. The results of the model calculations, together with experimental data underwent to the least-squares method statistical analysis.

5. Results and Discussions

Numerical simulations, of pressure changes for pneumatic transport of fly-ash in a long duct, are carried out based on the experimental conditions. The applied Euler-Euler approach offers an in-depth examination of the complex dynamics involved in pneumatic. Considering the two-phase nature of the system, it may provide important understanding of particle behavior, pressure drop, and other important aspects of pneumatic conveying.

Creating a mesh for Euler-Euler simulations in OpenFOAM can be a complex process. It involves defining the computational domain, generating a suitable mesh structure, and ensuring proper boundary conditions for both the gas and solid phases. Careful consideration is required to accurately represent the geometry and capture the relevant flow physics in pneumatic conveying systems.

The numerical model of two-phase flow is difficult to solve from a numerical point of view, with complex closure models. In long pipes, errors due to these closure models accumulate, and this can create numerical instability. Thus, only first two sections of pipeline 1 from [3, 4] were in focus of this study. The results of numerical simulations are presented in Fig. 3 and Fig. 4.

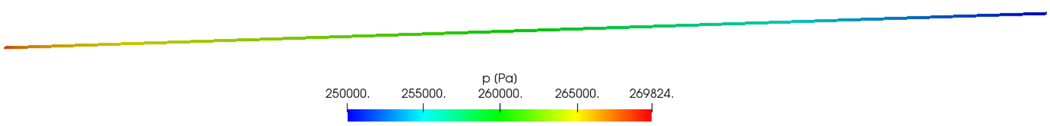

Figure 3. OpenFOAM numerical simulations results for pipeline section 1

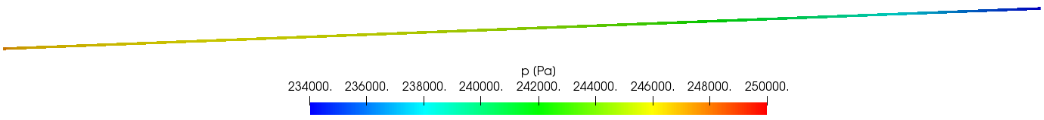

Figure 4. OpenFOAM numerical simulations results for pipeline section 2

On the basis of experimental studies of the operation on fly ash pneumatic transport, the results of the OpenFOAM pressure drop numerical simulation for pipeline 1 from [3, 4] are plotted against the experimental data and given in graphically, Fig. 5.

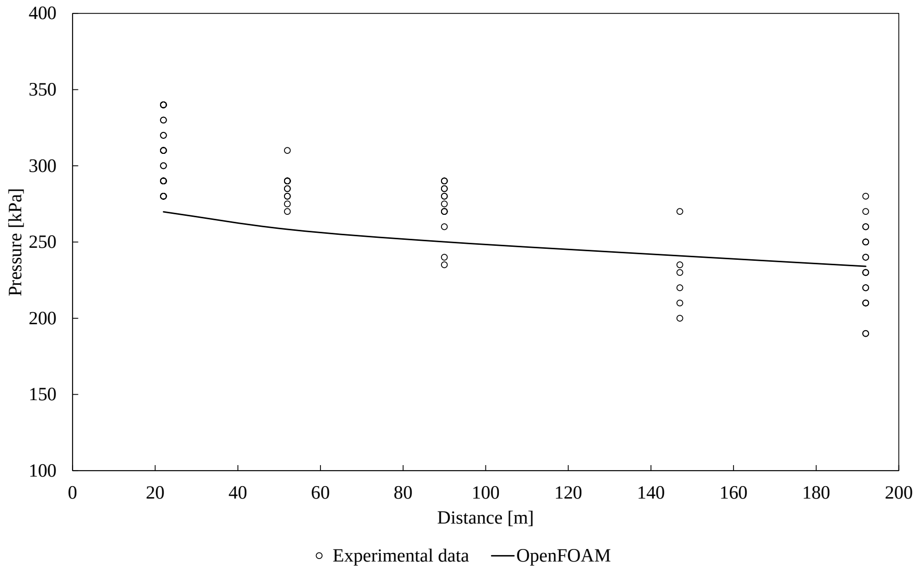

Figure 5. Measured pressure amplitudes and calculated pressure changes along the transport pipeline number 1 for Kolubara fly ash pneumatic conveying

The scatter in Fig. 5 indicates good agreement between the OpenFOAM numerical simulation results and the experimental data. The pressure values predicted using the OpenFOAM numerical model were found to slightly underestimate the experimental pressure values. The mean squared deviation between model output and data in validation dataset is determined to be 5.67%, although the correlation ratio was 63.97%. The reason might the fact that this study included only the drag force. Lift forces that cause particle redispersion in the pipe section are neglected. Considering that the true value of the restitution coefficient is unknown, its value was defined as e = 0.8. Accordingly, even a small discrepancy in this value, with respect to a large pipe length, can have a large impact on the numerical results.

Given that the phenomena in pneumatic transport are very diverse and complex [10], these finding should be considered very good. Therefore, this model may be considered acceptable for prediction of the fly ash pneumatic conveying capacity and pressure drop for the investigated pipeline. In future studies, the shadow-wall effect [49] for particle-wall interaction could be incorporated. This effect increases particle wall-normal velocity component and prevents particle deposition on the pipe wall.

6. Conclusion

Two large groups of numerical models are available to simulate pneumatic transport of solids in pipes, discrete element model and two-phase model. Long duct length, large number of particles and expensive computationally have led to using a two-phase model in this study. This was an especially interesting decision, since numerical studies of the pneumatic transport of solids through long inclined pipes using a two-phase model are scarce.

Numerical simulations of multiphase gas-solid flow, within a fly ash pneumatic conveying system, in 620 MW thermal power plant, are conducted and analyzed for such a flow configuration using the Euler-Euler approach and twoPhaseEulerFoam in OpenFOAM.

The main goal of this study is to conduct numerical simulations of the pneumatic conveying of lignite fly ash through stepped diameter, inclined, high-capacity long duct, based on real operating conditions in a 620 MW TPP unit. Also, the objective of this research is to verify the developed model by comparing the numerical results with experimental data of pressure measurements along the pipeline.

The numerical simulations input data are based on existing experimental data, and the results are obtained for the air mass flow rate 5500 Nm^3/h and the ash mass flow rate 77 t/h. Numerical mesh is generated as an O-grid for the two inclined telescopic pipeline sections, where every mesh cell is substantially larger than the diameter of the ash particle. First section has length of 90 m, diameter of 0.2604 m, and an inclination of 1.885 degree, the second has length of 102 m, diameter of 0.3097 m, and an inclination of 2.163 degree

This study found good agreement between the OpenFOAM numerical simulation results and the experimental data. The pressure values predicted using the OpenFOAM numerical model were found to slightly underestimate the experimental pressure values. The mean squared deviation between model output and data in validation dataset is determined to be 5.67% with correlation ratio 63.97%. The reason might be the fact that this study included only the drag force. Lift forces that cause particle redispersion in the pipe section are neglected. Considering that the true value of the restitution coefficient is unknown, its value was defined as $e = 0.8$. Accordingly, even a small discrepancy in this value, with respect to a large pipe length, can have a large impact on the numerical results.

Given that the phenomena in pneumatic transport are very diverse and complex, these finding should be considered very good, while this model may be considered acceptable for the analyzed conditions. The Euler-Euler model can be used in numerical simulations of dense pneumatic flows. The connection between length of pneumatic transport system and accuracy of conducted numerical computations could be further investigated. Experimental investigations are necessary in order to enable improvement of accuracy and stability of existing numerical models. At this point, Euler-Euler numerical simulations for calculation of pneumatic conveying need calibration and should be combined with experiments. Despite being complex in terms of required closure models, Euler-Euler model might be reliable for simulated conditions in future research. In future studies, the shadow-wall effect for particle-wall interaction could be incorporated.

It is expected that enormous progress will be achieved in resolving the issues with high-capacity and long-distance pneumatic conveying systems in the decades that follow. Thus, well-validated methodologies are a must, and one of the most reliable is comparisons to experimental data. That opens a new challenge in finding and establishing collaboration with experienced experimentalist and conducting extensive experiments that can be very challenging and demanding tasks. The alternative would be to use publicly available experimental data.

Acknowledgment

The presented results of the UB-FME here have been financed and supported by Ministry of Science, Technological Development and Innovation of the Republic of Serbia, contract No. 451-03-65/2024-03/200105 dated 5.2.2024.

References

[1] Birol, F., Key World energy statistics, IEA Publications, International Energy Agency, rue de la Federation, Paris, France

[2] EUROSTAT, Production Of Lignite In The Western Balkans, (2021), 2146, pp. 1-8

[3] Karličić, N. V, Svojstva pepela iz termoelektrana značajna za odlaganje i primenu (Characteristics of ash from thermal power plants important for disposal and utilization), Ph. D. thesis, University of Belgrade, Faculty of Mechanical Engineering, 2019

[4] Stevanovic, V.D., et al., Analysis Of Transient Ash Pneumatic Conveying Over Long Distance And Prediction Of Transport Capacity, *Powder Technol.*, *254* (2014), pp. 281-290

[5] Karlicic, N., et al., The Effect Of Permeability On Lignite Fly Ash Pneumatic Conveying System Design, *Rev. Chim.*, *69* (2018), 2, pp. 341-345

[6] Karličić, N., et al., Svojstva Pepela Koji Nastaje Sagorevanjem Lignita U Domaćim Termoelektranama Značajna Za Primenu U Industriji Građevinskog Materijala (FLY ASH PROPERTIES FROM LIGNITE COMBUSTION IN DOMESTIC POWER PLANTS IMPORTANT FOR USE IN BUILDING MATERIALS INDUSTRY), *Zb. Međunarodnog kongresa o procesnoj Ind. – Procesing; God. 27 Br. 1 27. Međunarodni Kongr. o procesnoj Ind.*, (2017)

[7] Deng, T., Bradley, M.S.A., Determination Of A Particle Size Distribution Criterion For Predicting Dense Phase Pneumatic Conveying Behaviour Of Granular And Powder Materials, *Powder Technol.*, *304* (2016), pp. 32-40

[8] Klinzing, G.E., A Review Of Pneumatic Conveying Status, Advances And Projections, *Powder Technol.*, *333* (2018), pp. 78-90

[9] Editorial, Experimental Validation, Anyone?, *Nat. Comput. Sci.*, *3* (2023), 5, pp. 361-361

[10] Karličić, N. V., et al., Validation Of Numerical Models For Prediction Of Pressure Drop In High Capacity Long Distance Lignite Fly Ash Pneumatic Conveying, *Therm. Sci.*, *27* (2023), 1, pp. 33-45

[11] Behera, N., et al., CFD Modeling And Analysis Of Dense Phase Pneumatic Conveying Of Fine Particles Including Particle Size Distribution, *Powder Technol.*, *244* (2013), pp. 30-37

[12] Miao, Z., et al., Numerical Simulation Of Dense-Phase Pneumatic Transport Of Powder In Horizontal Pipes, *Powder Technol.*, *361* (2020), pp. 62-73

[13] Pu, W., et al., Numerical Simulation On Dense Phase Pneumatic Conveying Of Pulverized Coal In Horizontal Pipe At High Pressure, *Chem. Eng. Sci.*, *65* (2010), 8, pp. 2500-2512

[14] Ariyaratne, W.K.H., et al., CFD Modeling Of Dilute Phase Pneumatic Conveying In A Horizontal Pipe Using Euler–Euler Approach, *Part. Sci. Technol.*, *37* (2019), 8, pp. 1015-1023

[15] Manjula, E.V.P.J., et al., A Review Of CFD Modelling Studies On Pneumatic Conveying And Challenges In Modelling Offshore Drill Cuttings Transport, *Powder Technol.*, *305* (2017), pp. 782-793

[16] Phillips, E.G., *Pneumatic Conveying: A Concise Treatment Of The Principles, Methods And Applications Of Pneumatic Conveyance Of Materials, With Special Reference To The Conveying And Elevating Of Heavy Solid Materials For Engineers, Works Managers, And Students*, Sir I. Pitman \& sons, Limited, 1921

[17] Ratnayake, C., A Comprehensive Scaling Up Technique for Pneumatic Transport Systems, Ph. D. thesis, The Norwegian University of Sceince and Technology (NTNU), 2005

[18] Mills, D., Pneumatic Conveying Design Guide, (2004), pp. 637

[19] Mittal, A., et al., An Investigation Into Pressure Fluctuations For Fluidized Dense-Phase Pneumatic Transport Of Fine Powders, *Powder Technol.*, *277* (2015), pp. 163-170

[20] Setia, G., et al., Modeling Solids Friction Factor For Fluidized Dense-Phase Pneumatic Transport Of Powders Using Two Layer Flow Theory, *Powder Technol.*, *294* (2016), pp. 80-92

[21] Wang, Y., et al., Gas–Solid Flow Behaviour Prediction For Sand In Bypass Pneumatic Conveying With Conventional Frictional-Kinetic Model, *Appl. Math. Model.*, *40* (2016), 23-24, pp. 9947-9965

[22] McGlinchey, D., *Characterisation Of Bulk Solids*, John Wiley \& Sons, 2009

[23] Chen, X., Wang, J., A Comparison Of Two-Fluid Model, Dense Discrete Particle Model And CFD-DEM Method For Modeling Impinging Gas-Solid Flows, *Powder Technol.*, *254* (2014), pp. 94-102

[24] Lu, H., et al., Experimental And CPFD Numerical Study On Hopper Discharge, *Ind. Eng. Chem. Res.*, *53* (2014), 30, pp. 12160-12169

[25] Ebrahimi, M., et al., Numerical And Experimental Study Of Horizontal Pneumatic Transportation Of Spherical And Low-Aspect-Ratio Cylindrical Particles, *Powder Technol.*, *293* (2016), pp. 48-59

[26] Bigda, J., CPFD Numerical Study Of Impact Dryer Performance, *Dry. Technol.*, *32* (2014), 11, pp. 1277-1288

[27] Fluent, A., others, ANSYS Fluent User's Guide, *Ansys Inc*, (2011)

[28] van Wachem, B., et al., Modelling Of Gas--Solid Turbulent Channel Flow With Non-Spherical Particles With Large Stokes Numbers, *Int. J. Multiph. Flow*, *68* (2015), pp. 80-92

[29] Zhao, F., et al., Four-Way Coupled Simulations Of Small Particles In Turbulent Channel Flow: The Effects Of Particle Shape And Stokes Number, *Phys. Fluids*, *27* (2015), 8

[30] Bagheri, G., Bonadonna, C., On The Drag Of Freely Falling Non-Spherical Particles, *Powder Technol.*, *301* (2016), pp. 526-544

[31] Kruger, R.A., 4 - Introduction To The Utilization Of Coal Combustion Products, in: *Coal Combustion Products (CCP's)* (Eds. T. Robl et al.), Woodhead Publishing, 2017, pp. 99-119

[32] Frye, L., Peukert, W., Attrition Of Bulk Solids In Pneumatic Conveying: Mechanisms And Material Properties, *Part. Sci. Technol.*, *20* (2002), 4, pp. 267-282

[33] Frye, L., Peukert, W., Progress In The Understanding Of Bulk Solids Attrition In Dilute Phase Pneumatic Conveying, *Powder Technol.*, *143* (2004), pp. 308-320

[34] Han, T., et al., DEM Simulation For Attrition Of Salt During Dilute-Phase Pneumatic Conveying, *Powder Technol.*, *129* (2003), 1-3, pp. 92-100

[35] Brosh, T., et al., DEM Simulation Of Particle Attrition In Dilute-Phase Pneumatic Conveying, *Granul. Matter*, *13* (2011), pp. 175-181

[36] Arastoopour, H., Numerical Simulation And Experimental Analysis Of Gas/Solid Flow Systems: 1999 Fluor-Daniel Plenary Lecture, *Powder Technol.*, *119* (2001), 2-3, pp. 59-67

[37] Ma, A., et al., Experimental and Numerical Research on Pressure Gradient of a Pneumatic Conveying Pipeline of Flyash, *Proceedings*, 2012 Second International Conference on Intelligent System Design and Engineering Application, January 2012, pp. 1247-1250

[38] Laín, S., Sommerfeld, M., Euler/Lagrange Computations Of Pneumatic Conveying In A Horizontal Channel With Different Wall Roughness, *Powder Technol.*, *184* (2008), 1, pp. 76-88

[39] Squires, K.D., Simonin, O., LES–DPS Of The Effect Of Wall Roughness On Dispersed-Phase Transport In Particle-Laden Turbulent Channel Flow, *Int. J. Heat Fluid Flow*, *27* (2006), 4, pp. 619-626

[40] Mallouppas, G., van Wachem, B., Large Eddy Simulations Of Turbulent Particle-Laden Channel Flow, *Int. J. Multiph. Flow*, *54* (2013), pp. 65-75

[41] van der Hoef, M.A., et al., Numerical Simulation Of Dense Gas-Solid Fluidized Beds: A Multiscale Modeling Strategy, *Annu. Rev. Fluid Mech.*, *40* (2008), 1, pp. 47-70

[42] Gidaspow, D., Multiphase Flow And Fluidization, *J. Fluid Mech.*, *287* (1995), pp. 405-407

[43] Ergun, S., Fluid Flow Through Packed Columns, *Chem. Eng. Prog.*, *48* (1952), pp. 89-94

[44] Wen, C.Y., Mechanics of fluidization, *Proceedings*, Fluid Particle Technology, Chem. Eng. Progress. Symposium Series, 1966, Vol. 62, pp. 100-111

[45] Ding, J., Gidaspow, D., A Bubbling Fluidization Model Using Kinetic Theory Of Granular Flow, *AIChE J.*, *36* (1990), 4, pp. 523-538

[46] Louge, M.Y., et al., The Role Of Particle Collisions In Pneumatic Transport, *J. Fluid Mech.*, *231* (1991), pp. 345-359

[47] Lun, C.K.K., et al., Kinetic Theories For Granular Flow: Inelastic Particles In Couette Flow And Slightly Inelastic Particles In A General Flowfield, *J. Fluid Mech.*, *140* (1984), pp. 223-256

[48] Sinclair, J.L., Jackson, R., Gas-particle Flow In A Vertical Pipe With Particle-particle Interactions, *AIChE J.*, *35* (1989), 9, pp. 1473-1486

[49] Sommerfeld, M., Huber, N., Experimental Analysis And Modelling Of Particle-Wall Collisions, *Int. J. Multiph. Flow*, *25* (1999), 6-7, pp. 1457-1489

Paper submitted: February 11, 2024

Paper revised: March 15, 2024

Paper accepted: May 22, 2024