

Pressure drop in horizontal positive dilute phase pneumatic teff grain conveyor

Lemi Demissie Boset^{1*}, Zewdu Abdi², Amana Wako³

(1. School of Mechanical and Automotive Engineering, Departments of Mechanical Engineering, Dilla University, P.O. Box 419, Dilla, Ethiopia;

Department of Agricultural and Biosystems Engineering, Faculty of Agriculture, University of Eswatini, P.O. Box 369 Luyengo, Eswatini; School of Mechanical, Chemical and Materials Engineering, Department of Agricultural Machinery Engineering, Adama Science and Technology University, P.O. Box 1888, Adama, Ethiopia)

Abstract: Pressure drop is one of the most important parameters affecting the performance and energy consumption of pneumatic conveying systems. This study experimentally and numerically investigated the pressure drop characteristics of a horizontal pipe section in a positive-pressure dilute-phase pneumatic conveyor for teff grain. The effects of pipe diameter $D = 46, 71, \text{ and } 102 \text{ mm}$, inlet air velocity $v = 12, 16, 18, 22, 25, \text{ and } 28 \text{ m s}^{-1}$, solid mass flow rate $\dot{m}_s = 0.009, 0.03, 0.044, 0.067, 0.084, \text{ and } 0.10 \text{ kg s}^{-1}$, and pipe length $L = 2000, 4000, \text{ and } 6000 \text{ mm}$ were evaluated. Experimental results were compared with CFD-DPM simulations and scale-up model predictions. The CFD-DPM model showed excellent agreement with the experimental data, achieving a coefficient of determination of $R^2 = 0.9987$. In addition, a strong correlation was observed between airflow rate and power losses associated with pressure drop. The findings demonstrate that pressure drop and the corresponding power requirements in horizontal pneumatic conveying of teff grain can be reliably predicted using both experimental and CFD-DPM approaches. The study provides useful design and scale-up information for improving the energy efficiency and operational performance of pneumatic conveying systems handling teff grain.

Keywords: CFD-DPM, pneumatic conveyor, pressure drop, pressure drop coefficient, scale-up model, teff grain.

Citation: Boset, L. D., Z. Abdi, and A. Wako. 2026. Pressure drop in horizontal positive dilute phase pneumatic teff grain conveyor. *Agricultural Engineering International: CIGR Journal*, 28(3):245-260.

Introduction

Teff is an important cereal crop indigenous to Ethiopia, with major diversity. A gluten-free grain with highly regarded nutritional benefits has recently attracted the attention of the modern food business. However, there isn't adequate information available on its handling and processing characteristics because it's a relatively new raw material for modern industry, it is the world's smallest grain, and because of its tiny, handling it without loss is challenging.

Pneumatic conveying system is chosen because it has a great advantage over mechanical conveyors, particularly for fine particle size dry bulk material. Pneumatic conveyor systems are vital when it comes to grain handling, which is widespread usage and the most optimal and sanitary method in most the food processing and grain handling industries (Özlü and Güner, 2022; Ebaid et al., 2016; Rahmanian Koushkaki et al., 2020).

Understanding the pneumatic conveying characteristics of materials is essential, as the pressure drop within the system largely determines them. Various factors affect this

Received date: 2025-01-18 Accepted date: 2026-07-14

*Corresponding author: Lemi Demissie Boset. School of Mechanical and Automotive Engineering, Departments of Mechanical Engineering, Dilla University. Email: lemidemissieboset@gmail.com.

pressure drop, including material properties, conveying velocities, pipe dimensions, and the design of a system (Subramaniam and Saiful Islam Wan Salim, 2024). By thoroughly examining these components, engineers can improve performance and decrease energy consumption in pneumatic conveying applications (Shijo and Behera, 2021; Kuang et al., 2020b; Ghafari and Sharifi, 2018).

In pneumatic conveyor, the pressure drop within these systems can be influenced by a multitude of factors, such as the characteristics of the conveyed material, the air velocity, and the overall system configuration (Kaur et al., 2017).

Theoretical models often approach the pressure drop as a combination of contributions from the gas flow, the dynamics of the solid particles, and the frictional losses resulting from the interactions between the gas and the solid particles.

Numerous studies have been conducted to investigate the pneumatic conveying characteristics of different dry fine bulk materials. However, for each specific material, a dedicated study is necessary to fully understand its unique characteristics (Zhu et al., 2022).

Determine pressure drop coefficient for a particular material is a crucial step in designing and optimizing the pneumatic conveyor system for its efficient operation.

In the case of teff grain, a specialized investigation is required to determine the pressure drop coefficient. This information is essential for the design and optimization of pneumatic conveyor systems tailored for the efficient handling and transportation of teff grain. By understanding the pressure drop characteristics of this specific material, engineers can ensure the pneumatic conveying system operates effectively, minimizing energy consumption and maximizing the overall performance of the system (Zhao and Su, 2010).

Dilute phase pneumatic conveying is a common technology in industrial mechanical and chemical applications. Numerical simulations were conducted to predict wall bounded particle flows and pressure decreases in dilute phase pneumatic (Shijo and Behera, 2021). CFD-DEM and CFD-DPM are common numerical approaches used in pneumatic conveyor studies (Reynoso Jardón et al., 2019).

A numerical study of gas-solid in complex flow for

three-dimensional systems using ANSYS Fluent showed good agreement between calculated and measured pressure gradients. Results indicated that pressure gradients increase with particle diameter and are influenced by particle density and roughness, with significant variations in drop pressure gradients in pneumatic conveyor (Kuang et al., 2020a).

The research examines losses due to bends in pneumatic conveying systems, emphasizing the importance of accurately estimating pressure drop in bends for predicting total pipeline pressure drop. Several existing models were compared with experimental data, revealing significant variations (Ma et al., 2010).

A new model for bend loss is proposed based on conveying data from various materials. DEM's ability to capture the complex behavior of particles, such as deformation, breakage, and fragmentation, makes it an attractive choice for studying the transport of polyethylene pellets in pneumatic conveyors (Sharma and Mallick, 2021). DPM is particularly suitable for simulating dilute particle flows, offering fast computations and reduced computer resource requirements. DPM's simplicity and efficiency make it an attractive choice for studying pneumatic conveyors, especially when dealing with large-scale systems or when a high degree of accuracy is not required (Singh et al., 2023).

Two-fluid model for gas-solid two-phase flow in a vertical riser, using a conservative variable approach for the gas phase and the fourth-order Runge-Kutta method for the solid phase, model predictions for pressure drop align well with experimental data and Eulerian-Lagrangian predictions. Related How does the conservative variable approach improve model accuracy What are the advantages of using the fourth-order Runge-Kutta method for solid phase modeling How does the Eulerian-Lagrangian method compare to the two-fluid model in predicting pressure drop What factors contribute to the increase in minimum pressure drop velocity How does system total pressure influence the minimum pressure drop velocity (Roberts et al., 2022).

The research finding named using a coupled CFD - DPM approach to predict particle settling in a horizontal air stream is studied by both experiment and numerical CFD-DPM simulation, the finding was first

experimentally evaluated and CFD coupled with DPM simulation was employed to predict particle deposition in the horizontal chamber. The results show a good agreement between experiments and CFD – DPM results (El-Behery et al., 2017).

The study presented a method for the rapid optimization design of the pneumatic conveying system of a hybrid rice pollen collection unit. The results might give a reference for the optimization design of the pneumatic conveying system of a hybrid rice pollen collection unit (Stone et al., 2019). The researchers mention that fluctuating gas velocity is hard to predict with the currently available turbulence modulation modeling choices, implying more efforts are needed to understand the turbulence in the future (Mezhericher et al., 2011).

The work on Predicting Horizontal Pneumatic Conveying of Large Coal Particles Using a Discrete Phase Model focuses on large particles (>5 mm), as there are few studies on such sizes. To forecast the horizontal pneumatic conveyance of big coal particles, coupling methods based on the Euler-Lagrange approach and discrete phase model (DPM) were utilized in the simulated study. When compared to the experimental results under the same working conditions, the particle trajectory obtained by simulation is similar to the particle distribution at the same position in the experiment, indicating that the simulation method is suitable for horizontal pneumatic conveying of large particles. The effects of particle size, flow field velocity, solid-gas rate, and pipe diameter on the wall abrasion during horizontal pneumatic conveying are also investigated using multifactor simulations, which provide a simulation reference and design guidelines for the pneumatic conveying of large particles (Huang et al., 2021).

DPM can be utilized for modeling particle pneumatic conveying, particularly for analyzing the behavior of particles in a pneumatic conveyor system. Researchers where shown that DPM is more appropriate for modeling dilute particle flow (Dilute phase pneumatic conveyor), where the particle volume fraction is less than (10%) (El-Behery et al., 2017). The DPM can be used to analyze various factors such as particle trajectories, deposition, and optimum conveying velocity in pneumatic conveying systems (Stone et al., 2019). Different research findings

indicate DPM is valuable for the design and optimization of pneumatic conveying systems, as it provides insights into the behavior of particles under different conditions, along with the Euler-Lagrange (CFD-DPM) method, is used to analyze the flow pattern and make predictions that can be beneficial for system design and the selection of the optimum conveying velocity.

This study is novel in its focus on teff grain, a fine bulk material with unique handling challenges due to its small size and lightweight nature, which has been largely overlooked in pneumatic conveying research. By integrating experimental validation with CFD-DPM simulation, it employs an innovative dual-methodology to precisely quantify and analyse pressure drops in horizontal pneumatic conveying systems. This approach not only ensures reliability but also provides new insights into the flow dynamics of teff grain, addressing a critical gap in the field.

2 Materials and methods

2.1 Experimental set up

Eight new varieties of teff were collected according to the year of released as shown in Figure 1 (a) Kora, (b) Boset, (c) Dagim, (d) Felagot, (e) Bora, (f) Ebba, (g) Bishoftu, and (h) Boni were obtained from Debre Zeit Agricultural Research Center of the Institute of Agricultural Research in Ethiopia (EIAR). The Ethiopian Institute of Agricultural Research (EIAR)'s National teff Improvement Program distributed such teff species. The laboratory tests were conducted at Adama Science and Technology University, Chemical Department Laboratory.

Moisture content of eight different types of teff grain measured according to ASAE standards S3523 (ASAE, 1994). Recorded moisture content for each variety of teff grain is 9.14%-15%, 7.73%-13.48%, 10.14%-14.35%, 9.5%-15.28%, 11.59%-14.14%, 9.5%-13.36%, 10.25%-12.36%, and 8.65%-11.96% for Felagot, Ebba, Bishoftu, Dagim, Boni, Boset, Kora, and Bora, respectively. Teff grain SEM measurements have been obtained for eight samples, including small, medium, and big grains. The measurements showed a wide range of length, width, and thickness; the smallest grain was 476 μm , while the largest was 1.1 mm for measured eight teff grain size. Grain dimensions showed considerable variety, with

intermediate sizes including grains measuring 539 μm , 681 μm , 733 μm , 864 μm , 936 μm , and 1 mm as shown in Figure 2.

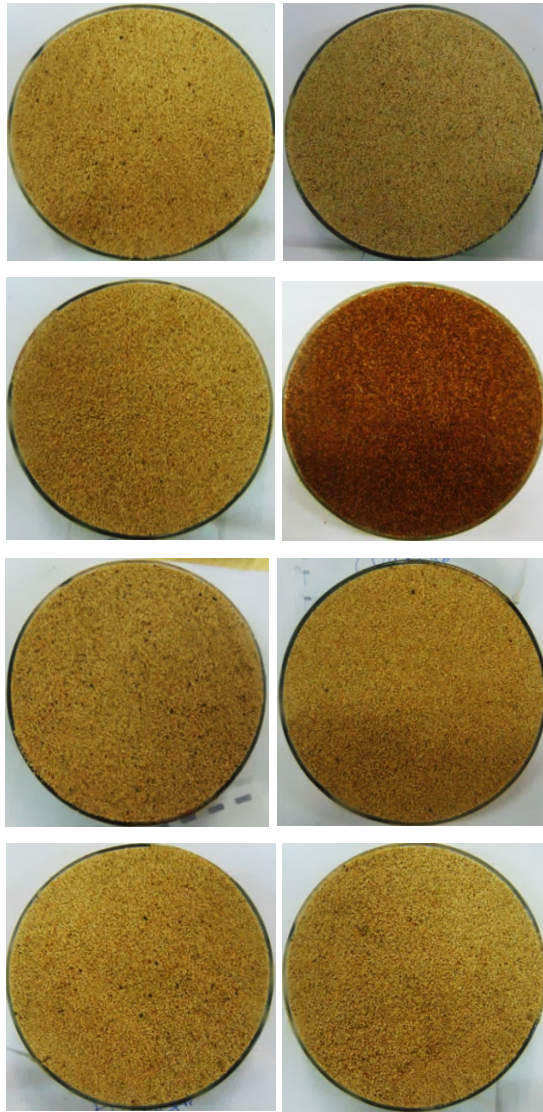


Figure 1 Different varieties of teff grain

Geometric mean diameter $D_g(\text{mm})$ and Arithmetic mean diameter $D_a(\text{mm})$ calculated using the following Equation (1) and (2).

$$D_g = \sqrt[3]{W \times T \times L} \quad (1)$$

$$D_a = \frac{W + T + L}{3} \quad (2)$$

Where,

L is length (mm);

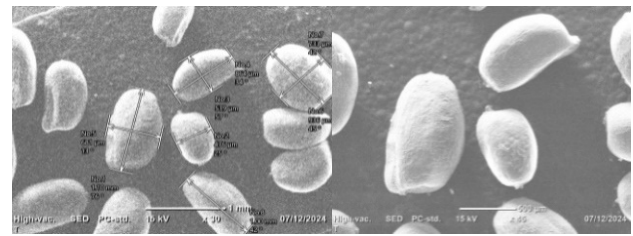
W is width (mm);

T is thickness (mm), is presented in Table 1.

The study consider practical application in local for feeding teff milling machine, small scale teff grain milling machine its capacity is 300kg h^{-1} -500kg h^{-1} which is mostly used in local milling house, therefore (m_s) for

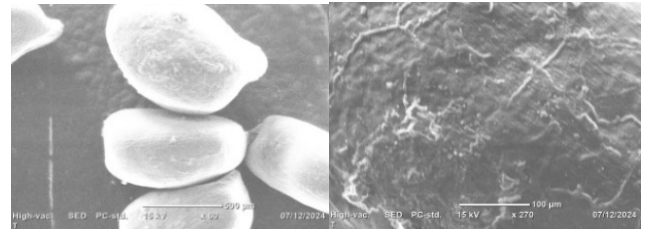
desired dilute phase pneumatic conveying system is 360kg h^{-1} or 0.1kg s^{-1} . The study used blower with air flow rate 19m³ min^{-1} . From the eight teff grain varieties evaluated, three varieties such as Felagot (DZ-Cr-442RIL77C), Bishoftu (DZ-Cr-497RIL133), and Boni (DZ-Cr-498RIL37) were selected for this study based on their distinct characteristics, including varietal differences, grain color, and year of release. A solid loading ratio, or phase density, is a useful parameter in helping to visualize the flow. It is $m_s(\text{kg s}^{-1}) / m_a(\text{kg s}^{-1})$. Maximum recommended solid loading ratio for dilute phase pneumatic conveyor is 15 (Ratnayake et al., 2007). μ is solid loading ratio (-).

$$\mu = \frac{m_s}{m_a} \quad (3)$$



(a) SEM image X30

(b) SEM image X45



(c) SEM image X60

(d) SEM image X60

Figure 2 SEM image

According to Ratnayake et al. (2007), the Rizk to predict the gas velocity at saltation using calculated solids loading ratio mass flow rate using Equation (4).

$$\mu = \frac{1}{10^\delta} \left(\frac{U_o}{(gD)^{1/2}} \right)^\chi \quad (4)$$

Where,

χ and δ are constants: $\chi = 1.1d_p + 2.5$, $\delta = 1.44d_p + 1.96$.

The value of χ and δ are constants equal to 3.215 and 2.896 respectively, g is gravitational acceleration (m s^{-2}) and D is pipe diameter (mm); (design air velocity is must be greater than saltation velocity, the design velocity is superficial gas velocity v (m s^{-1}) is calculated from saltation velocity U_o (m s^{-1}) as:

$$v = 1.5U_o \quad (5)$$

Based on the Rizk method, the minimum saltation velocity for teff grain was estimated to be 12 m s⁻¹. Accordingly, six inlet air velocities (12, 16, 18, 22, 25, and 28 m s⁻¹) were selected to cover the operating range of the positive-pressure dilute-phase pneumatic conveying system, extending from the predicted minimum saltation velocity to velocities well above it. These operating conditions were chosen to evaluate the effect of airflow on the conveying characteristics of teff grain, including pressure

drop, conveying efficiency, particle velocity, and saltation behavior.

The selected inlet air velocities were applied to all three teff varieties under identical experimental conditions. For each inlet air velocity, the corresponding air mass flow rate was calculated as 0.024, 0.032, 0.036, 0.045, 0.051, and 0.057 kg s⁻¹, respectively, based on the pipe cross-sectional area and the density of air. These operating parameters were used to conduct the laboratory experiments, as illustrated in Figure 4.

Table 1 Summary of measured teff grain size

Teff Variety	Width (mm)	Thickness (mm)	Length (mm)
Boset	0.63±0.05	0.56±0.10	1.01±0.13
Kora	0.68±0.10	0.57±0.06	1.09±0.18
Dagim	0.67±0.08	0.61±0.07	1.17±0.17
Felagot	0.52±0.07	0.52±0.08	0.92±0.08
Bora	0.58±0.09	0.56±0.04	1.10±0.18
Ebba	0.70±0.08	0.58±0.12	1.09±0.15
Bishoftu	0.59±0.14	0.54±0.08	0.94±0.15
Boni	0.59±0.09	0.53±0.03	0.94±0.15

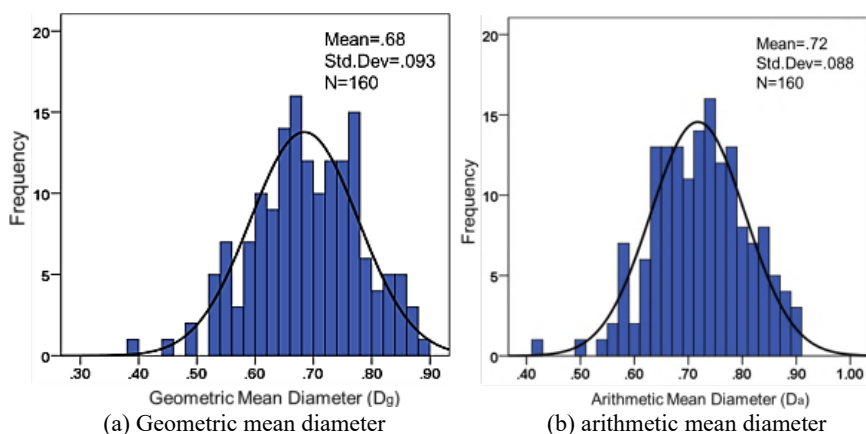
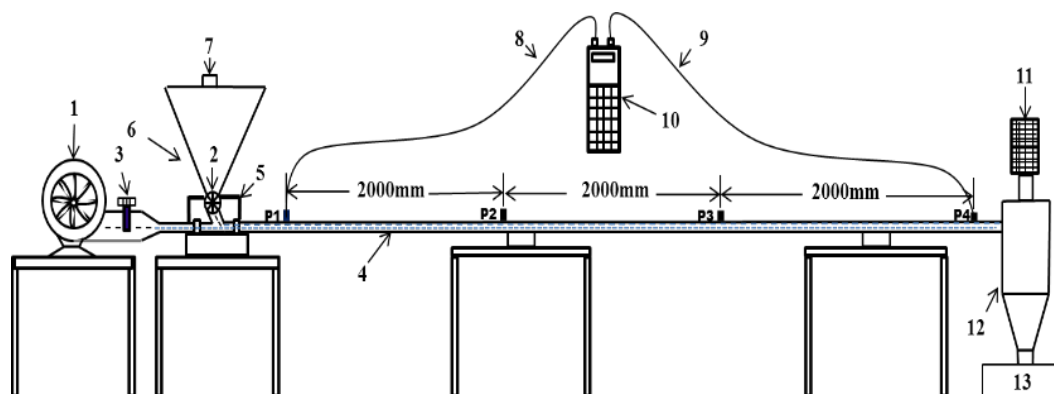


Figure 3 Geometric and arithmetic mean diameter



1: blower; 2: rotary feeder; 3: Air flow control valve; 4: pipe line; 5: material flow control device; 6: hopper; 7: material inlet; 8 and 9: hose; 10: pressure transducer; 11: air filter; 12: cyclone separator; and 13: material collector

Figure 4 Horizontal Pneumatic conveyor Experimental set up

Figure 5 illustrates the basic components of the laboratory test setup, providing a clear visual representation of the experimental system.



(a) blower



(b) pipe line and cyclone separator



(c) air flow control and feeding system



(d) digital manometer



(e) digital anemometer

Figure 5 Basic components of the laboratory test setup

2.2 Numerical analysis

In CFD-DPM simulation, number iteration depends up on scaled residuals. When scaled residuals approach to zero and less than $(1E^{-5})$ – $(1E^{-6})$ the simulation became converged (Cao et al., 2022). The equations used to compute the trajectory of the discrete phase particle is given as follows:

$$\frac{du_p}{dt} = F_d(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} + F \quad (6)$$

Where,

u_p ($m\ s^{-1}$) and u ($m\ s^{-1}$) are the particle and fluid velocities, respectively;

ρ_p ($kg\ m^{-3}$), and ρ ($kg\ m^{-3}$) are the particle and fluid

densities, respectively. The drag force, F_d (N), acting on a spherical particle is:

$$F_d = \frac{18\mu C_D Re}{\rho_p d_p^2 24} \quad (7)$$

Where,

Re is the Reynolds number;

μ is the dynamic viscosity of the fluid;

C_D is the drag coefficient based on the shape of the particle (Asefa et al., 2023).

2.3 Turbulence model

The flow regime in the pneumatic conveying pipeline was evaluated using the Reynolds number, calculated from the inlet air velocity, pipe diameter, air density, and dynamic viscosity. For the selected inlet air velocities of

12, 16, 18, 22, 25, and 28 m s⁻¹, the corresponding Reynolds numbers were all well above 2300, indicating fully turbulent flow throughout the conveying system. The value of 2300 is the widely accepted critical Reynolds number that marks the transition from laminar to turbulent flow in circular pipes. Because the calculated Reynolds numbers exceeded this critical value, turbulence effects were considered in the numerical analysis. Therefore, the governing mass and momentum transport equations were formulated in their Reynolds-averaged (RANS) form to accurately account for turbulent flow behavior in the pneumatic conveying system (Cao et al., 2022). The mass and momentum transport equations have the following Reynolds- and time-averaged forms:

$$\frac{\partial \rho}{\partial t} = \nabla \cdot (\rho \mathbf{v}) = S_m \quad (8)$$

$$\frac{\partial \rho}{\partial t} + \nabla \rho (\mathbf{v} \cdot \mathbf{v}) = \nabla \cdot \mathbf{p} + \nabla \bar{T} + \nabla \rho (\mathbf{v}' \cdot \mathbf{v}') + \rho \mathbf{g} + \mathbf{F} \quad (9)$$

In this study, the Renormalization Group (RNG) k-ε turbulence model, an enhanced variant of the standard k-ε model, was employed to simulate the turbulent airflow in the pneumatic conveying system. The RNG k-ε model is derived by applying the renormalization group statistical method to the instantaneous Navier–Stokes equations. Compared with the standard k-ε model, it incorporates an additional term in the turbulent dissipation (ε) transport equation, accounts for the effects of rapidly strained and swirling flows, and uses analytically derived turbulent Prandtl numbers instead of constant values. These modifications improve the prediction of complex turbulent flows commonly encountered in pneumatic conveying systems. Therefore, the RNG k-ε model is generally considered more accurate and reliable than the standard k-ε model for simulations involving strong streamline curvature, flow separation, and high strain rates (Wang et al., 2023). The RNG k-ε model is generally considered more accurate and reliable than the standard k-ε model because it incorporates additional corrections for rapidly strained and swirling flows, leading to improved predictions of complex turbulent flow fields. Although the RNG k-ε model differs in its formulation, it is still based on the k-ε turbulence framework. Therefore, the transport equations for the turbulent kinetic energy (k) and turbulent dissipation rate (ε) are solved using the RNG-modified

formulations rather than the standard k-ε equations. Accordingly, the governing transport equations presented below correspond to the RNG k-ε model adopted in this study (Dutta et al., 2016).

$$\frac{\partial \rho k}{\partial t} + \nabla \rho (k \cdot \mathbf{v}) = \nabla \cdot (\mu + \frac{\mu_t}{\sigma_k} \nabla k) + G_k + G_b + \rho \varepsilon + Y_M + S_k \quad (10)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \nabla \rho (\varepsilon \cdot \mathbf{v}) = \nabla \cdot (\mu + \frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon) + C_{1\varepsilon} \frac{\varepsilon}{k} + (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (11)$$

Where,

Y_M accounts for compressibility effects on turbulence;

G_k is the generation of turbulence due to the velocity gradients (local strain);

G_b is the generation of turbulence due to buoyancy effects;

σ_k and σ_ε are the turbulent Prandtl numbers for k and ε.

$C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ and in the dissipation rate equation are constants that are derived by calibration with experiments;

S_k and S_ε are user defined source terms.

The local solution for k - ε enables calculation of the local eddy viscosity, which takes the form: $\mu_t = \rho C_\mu k^2 / \varepsilon$ Singh et al. (2023). The eddy viscosity appears in the momentum equations and in the k and ε equations so the solution procedure is strongly coupled.

The eddy viscosity appears in both the Reynolds-averaged momentum equations and the turbulent dissipation (ε) equation, creating a strong coupling between the velocity field and the turbulence variables (k and ε). Therefore, changes in the flow field affect the turbulence quantities, which in turn influence the momentum equations. This coupled behavior requires an iterative solution procedure to achieve convergence and accurately predict the turbulent airflow characteristics in the pneumatic conveying system (Arriaga et al., 2025; Singh et al., 2023).

2. 4 CFD-DPM ANSYS Fluent simulation setup

Teff grain is a non-spherical particle with a natural size distribution. In this study, a shape factor of 0.76 was used to represent its non-spherical geometry in the CFD-DPM simulation and the Rosin-Rammler particle diameter distribution model. To capture the turbulent flow characteristics, the RNG k-ε turbulence model was employed in this study. The RNG k-ε model is an enhanced version of the standard k-ε model, incorporating

additional corrections for rapidly strained and swirling flows commonly encountered in pneumatic conveying systems. These improvements provide more accurate predictions of airflow and particle transport than the standard $k-\epsilon$ model. For pressure–velocity coupling, the SIMPLE algorithm with a pressure-based solver was used to solve the governing flow equations efficiently (Roberts et al., 2022). The discretization of momentum, turbulence kinetic energy, and turbulence dissipation rate equations was done using second-order upwind schemes (Yusop et al., 2023).

The pressure drop across the pneumatic conveyor was computed using the CFD results from the surface integral result for area-weighted average static pressure and pressure drop calculated from the difference between static pressure at the inlet and outlet. After completing a simulation that meets convergence criteria, particle data can be exported to CFD Post. This powerful tool allows for detailed visualization of pressure distribution and particle velocity profiles, enhancing the understanding of fluid dynamics and improving system optimization (Xu et al., 2024).

The CFD-DPM setup involves several key components and parameters essential for simulating particle dynamics within a fluid medium using ANSYS software. The geometry modeling is performed using ANSYS SpaceClaim 2024R1 to create a 3D model, with mesh generation ensuring a mesh size to particle mean diameter ratio greater than 4, specifically set to 4.5 mm. Particle properties include an effective grain mean diameter (d_p) of 0.68 mm, with minimum and maximum diameters of 0.54 mm and 0.82 mm, respectively. Mass flow rates for the particles are set at 0.009 kg s^{-1} , 0.03 kg s^{-1} , 0.044 kg s^{-1} , and 0.067 kg s^{-1} , with a shape factor of 0.76 for non-spherical particles and a density (ρ_p) of 1120 kg m^{-3} . The fluid properties are defined with air density (ρ_a) at 1.225 kg m^{-3} and viscosity at $1.7894\text{E-}5 \text{ kg m}^{-1}\text{s}^{-1}$. Operating conditions include a gravitational acceleration (g) of -9.81 m s^{-2} along the y-axis.

In this study, the CFD-DPM simulation employed two-way coupling to account for the interaction between the continuous air phase and discrete teff grain particles. The RNG $k-\epsilon$ turbulence model with standard wall functions was used to model turbulent flow, while surface injection

and the Rosin-Rammler particle size distribution were adopted to represent particle injection and size variation. The inlet air velocity was specified between 12 and 28 m s^{-1} , with an outlet pressure of 0 Pa and escape boundary conditions for both air and particles at the outlet. The governing equations were solved using the SIMPLE pressure–velocity coupling algorithm and second-order discretization schemes. Simulations were run for 100–500 iterations until the convergence criteria for continuity, momentum, turbulence quantities, and the discrete phase were satisfied. Area-weighted average static pressure together with pressure and velocity distributions were extracted for post-processing.

Determination of pressure drop in horizontal pneumatic conveyor using CFD simulation typically employs a two-way coupling approach, which accounts for the interaction between particles and the continuous phase. In this context, the boundary condition for the walls is set to reflect. This reflect boundary condition means that when a particle collides with a solid surface, it is reflected into the computational domain rather than being absorbed or passing through the boundary.

In the present study, the recommended mesh requirements for CFD-DPM simulations using the $k-\epsilon$ turbulence model were applied. The mesh size was selected to be at least four times larger than the teff grain particle size, ensuring that the computational mesh was sufficiently coarse to accurately capture particle motion while adequately representing the bulk flow characteristics (Xu et al., 2024). In addition, the non-dimensional wall distance (Y^+) was monitored and maintained within the recommended range of 30–300. This range corresponds to the logarithmic region of the boundary layer, where the standard $k-\epsilon$ turbulence model is applicable and provides reliable predictions of turbulent flow behavior near solid boundaries (Dutta and Nandi, 2019). The selected mesh size and Y^+ distribution obtained in the present CFD-DPM simulation are presented in Figure 6.

Mesh sensitivity analysis was also performed to ensure grid independent results and accurate prediction of pressure drop in the horizontal pneumatic conveying system. Several mesh configurations with different levels of refinement were generated and evaluated using CFD-

DPM simulations. The predicted pressure drop values were compared as the mesh resolution increased, and the final mesh was selected when further refinement resulted in negligible changes in pressure drop, confirming mesh independence (Ratnayake et al., 2007). The relationship between mesh size and pressure drop, together with the corresponding Y^+ distribution, is shown in Figure 6(a) and Figure 6(b), respectively. Therefore, the selected mesh configuration provided an appropriate balance between computational efficiency and numerical accuracy for the CFD-DPM analysis.

The pressure drop curve is refined until it either forms a flat plateau or becomes invariant. In the CFD-DPM simulation, the mesh size is set to be at least four times

larger than the particle size, as the mean diameter of the teff grain is 0.68 mm. The mesh size range for this investigation starts from the finest mesh size of 2.72 mm to the coarsest mesh size of 10 mm.

The figure 6 (a) is indicated that pressure drop curve having a flat section, which shows that more mesh refinement, has no effect on the solution. The mesh refinement flat curve procedure ranges from 6.25 mm to 2.72 mm with a midpoint value of 4.5 mm traced and corresponding Y^+ value of 97.16 has been obtained, Figure 6 (b) Y^+ for mesh size = 4.5mm. In order to guarantee precision and uniformity in the CFD-DPM analysis, a mesh size of 4.5 mm is chosen for every following simulation.

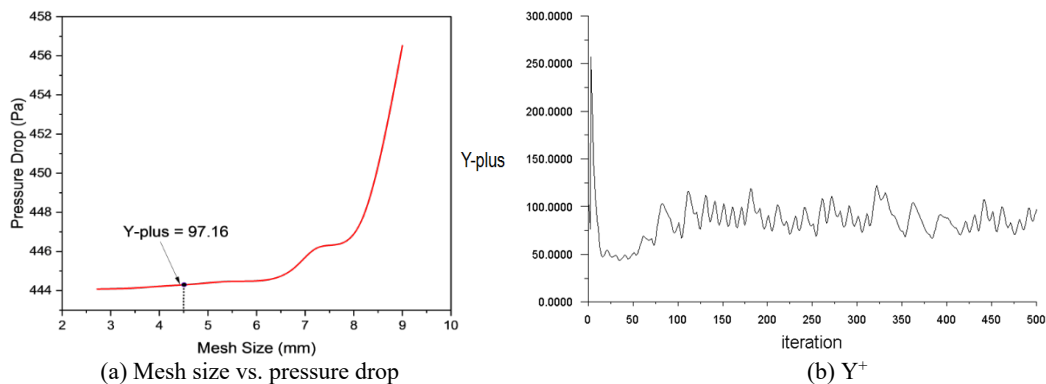


Figure 6 Mesh size vs. pressure drop curve

Table 2 Summarized the boundary conditions for CFD-DPM simulation in a horizontal positive dilute-phase pneumatic conveyor

Parameter	Details
Particle Shape and Size	Teff grain is non-spherical with a shape factor of 0.76. Effective mean diameter: 0.68 mm, minimum diameter: 0.54 mm, maximum diameter: 0.82 mm.
Particle Injection	Rosin-Rammler particle diameter distribution. surface injection type. Mass flow rates: 0.009 kg s ⁻¹ , 0.03 kg s ⁻¹ , 0.044 kg s ⁻¹ , 0.067 kg s ⁻¹ .
Particle Properties	Density (ρ_p): 1120 kg m ⁻³ .
Fluid Properties	Air density (ρ_a): 1.225 kg m ⁻³ , Viscosity: 1.7894E ⁻⁵ kg m ⁻¹ s ⁻¹ .
Boundary Conditions	Air velocities at the inlet: 12 m s ⁻¹ to 28 m s ⁻¹ along the Z-axis. Outlet pressure: 0 Pa. Reflect condition at walls.
Operating Conditions	Gravitational acceleration: -9.81 m s ⁻² along the y-axis.
Mesh Settings	Mesh size to particle mean diameter ratio: >4. Range: 2.72 mm (fine mesh) to 10 mm (coarse mesh).
Turbulence Model	k- ϵ RNG with standard wall functions. Maintain non dimensional wall distance (Y^+): 30-300.
Solver and Discretization	Pressure-velocity coupling using SIMPLE. Second-order upwind schemes for pressure, momentum, turbulence kinetic energy, and dissipation rate.
Convergence Criteria	Residuals for continuity, velocities, turbulence quantities, and discrete phase at specific thresholds. Iterations: 100 to 500.
Mesh Sensitivity	Conduct meshes sensitivity analysis to ensure grid independence. Refine mesh until pressure drop curve stabilizes.
Post-Processing	Export particle data to CFD Post for detailed visualization of pressure distribution and particle velocity profiles.
Pressure Drop Calculation	Computed from area-weighted average static pressure difference between inlet and outlet.

2.5 Scale up model

Modeling technique for designing pneumatic conveying systems by addressing the whole pipeline together with all accessories, one method that is used for predicting pressure drop coefficient involves testing a sample of the product to be conveyed in the final system in a pilot scale rig over a wide range of operating conditions, and measuring the product and air flow rates and resulting pressure drops. The obtained data are then scaled by experimentally determined factors to predict the pressure drop in the full-scale system. The scaling of test data is considered to be one of the most important stages of the design process, in that it provides the link between laboratory-scale apparatus and full-scale industrial installations. Hence, the accuracy and reliability of scale-up are essential. There are two approaches in scaling up techniques available in literature; the Global Approach and the Piecewise Approach. Both approaches have their advantages and undesirable characteristics as well. The basic condition of the technique is that conveying conditions, in terms of air velocities, should be identical for the pilot plant and full-scale plant. It has been revealed that the equations used in Mills scale-up technique are inadequate, especially when data are scaled up concerning pipeline diameter (Datta and Ratnayaka, 2003). They proposed following scale-up equation:

$$m_{s2} = m_{s1} \left(\frac{L_1}{L_2} \left(\frac{D_2}{D_1} \right) \right)^{2.8} \quad (12)$$

Where,

m_{s2} = solid flow rate (kg m^{-3});

m_{s1} = solid flow rate (kg m^{-3});

L_1 , D_1 , L_2 , D_1 and D_2 were pipe lengths (m) and diameters (m).

$$\Delta P = 4 \frac{f \rho_a v^2 L}{2D} \quad (13)$$

Where,

ΔP = pressure drop (Pa);

f = air friction factor (-);

ρ_a = air density (kg m^{-3});

v = air velocity (m s^{-1});

L = pipe length (m);

D = pipe diameter (m).

$$\Delta p_{st} = \frac{1}{2} K_{st} \rho_{sus} v_{entry}^2 \frac{\Delta L}{D} \quad (14)$$

Where,

Δp_{st} = pressure drop (Pa);

v_{entry} = air velocity at entry (m s^{-1});

K_{st} = pressure drop coefficient (-);

ρ_{sus} = suspension density (kg m^{-3});

ΔL = change in pipe length (m), and D = pipe diameter (m).

It should be highlighted that all v_{entry} value is the true gas velocity at the entry section of the concerned pipe section or pipe component, K_b is pressure drop coefficient for bend pipeline, and suspension density ρ_{sus} can be defined as the mixture density when a short pipe element is considered. As an equation, it can be presented in the following way (Xu et al., 2024).

$$\rho_{sus} = \frac{m_s + m_a}{V_s + V_a} \quad (15)$$

Where,

ρ_{sus} = suspension density (kg m^{-3});

m_s = solid mass flow rate (kg s^{-1});

m_a = air mass flow rate (kg s^{-1});

V_s = solid volume (m^{-3}), and V_a = volume of air (m^{-3}).

3 Results and discussions

3.1 Pressure drop in horizontal positive dilute phase pneumatic teff grain conveyor

The influence of solid mass flow rate and inlet air velocity on the pressure drop characteristics of the pneumatic Teff grain conveying system is presented in Figure 7. The results reveal a significant relationship between these operating parameters and the pressure drop behavior of the system. As the solid mass flow rate increases, a greater quantity of teff grains is introduced into the conveying pipeline, which intensifies particle-air interactions and increases resistance to airflow. Consequently, a higher pressure gradient is required to provide the additional momentum needed for particle acceleration and transportation, resulting in an increase in pressure drop. Similarly, increasing the inlet air velocity enhances the conveying capacity by generating greater aerodynamic drag force on the grains and improving

particle suspension; however, it also increases frictional losses, turbulence intensity, and energy dissipation within the conveying system, thereby contributing to a further rise in pressure drop. Therefore, while higher air velocity is necessary to maintain stable particle transport,

excessive pressure drop may reduce conveying efficiency, increase energy consumption, and lead to flow instability, particle accumulation, or potential blockage in the pneumatic conveying system.

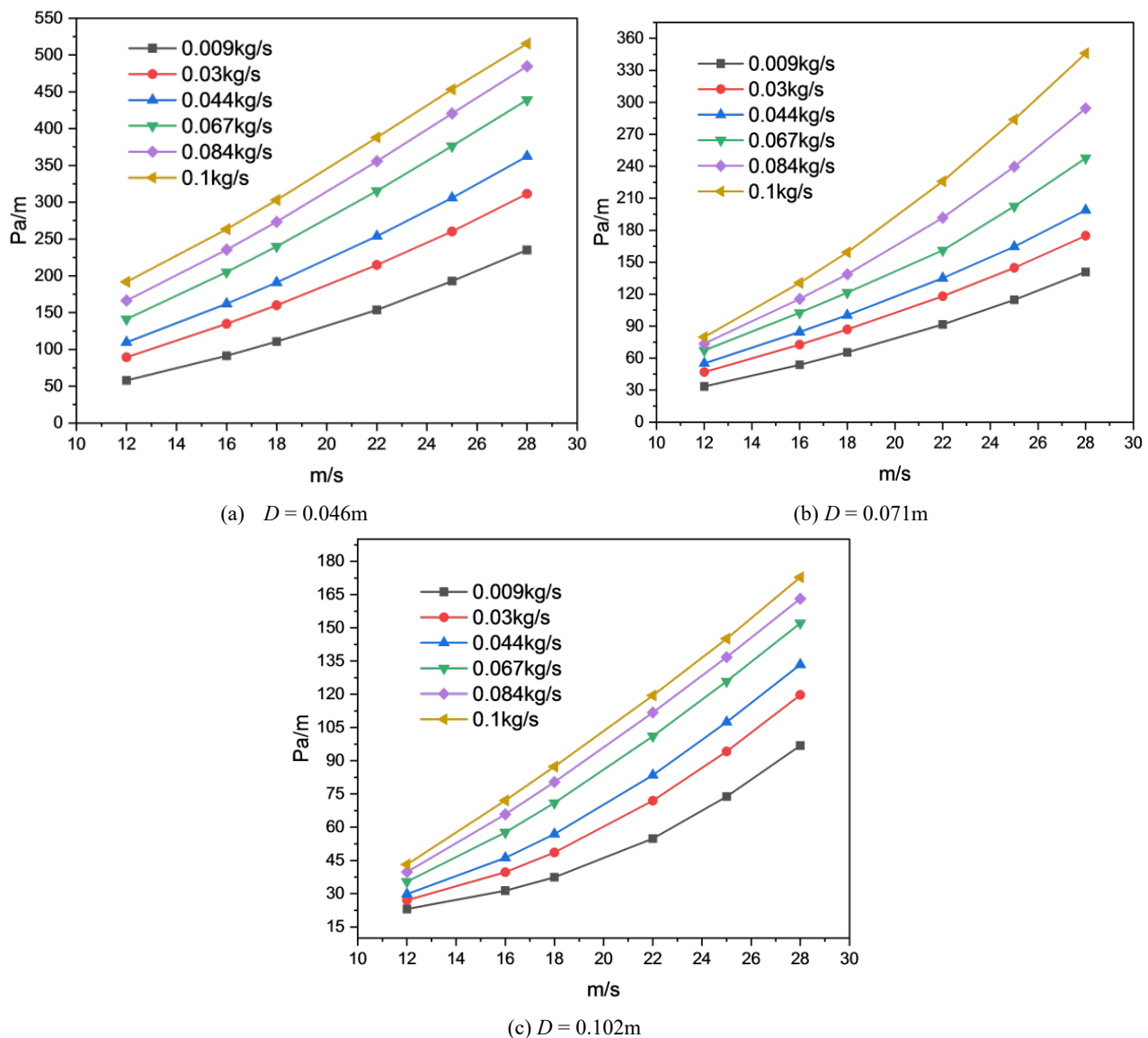


Figure 7 Pressure drop vs. inlet air velocity

3.2 Power loss due to pressure drop

Figure 8 highlight that as the inlet air velocity in a pneumatic teff grain conveyor increases, there is a corresponding rise in pressure drop across the system, and this pressure drop is a critical factor that affects the overall efficiency of the conveyor, as it directly correlates with power loss. When the air velocity is heightened, the resistance encountered by the air flow increases, leading to a more significant pressure drop. Consequently, this escalation in pressure drop results in greater energy consumption, as system must work harder to maintain the

necessary air flow rate. This relationship highlights the importance of optimizing inlet air velocity to balance performance and energy efficiency.

Figure 9 illustrate a strong correlation between the square of inlet air velocity and the pressure drop coefficient in pneumatic teff grain conveyors. As the square of inlet air velocity increases, the pressure drop coefficient decreases, indicating an inverse relationship. This finding allows us to effectively determine pressure drop coefficient for teff grain without needing to account for additional variables. Such insights are crucial for

optimizing system design and improving operational

efficiency in grain conveying processes.

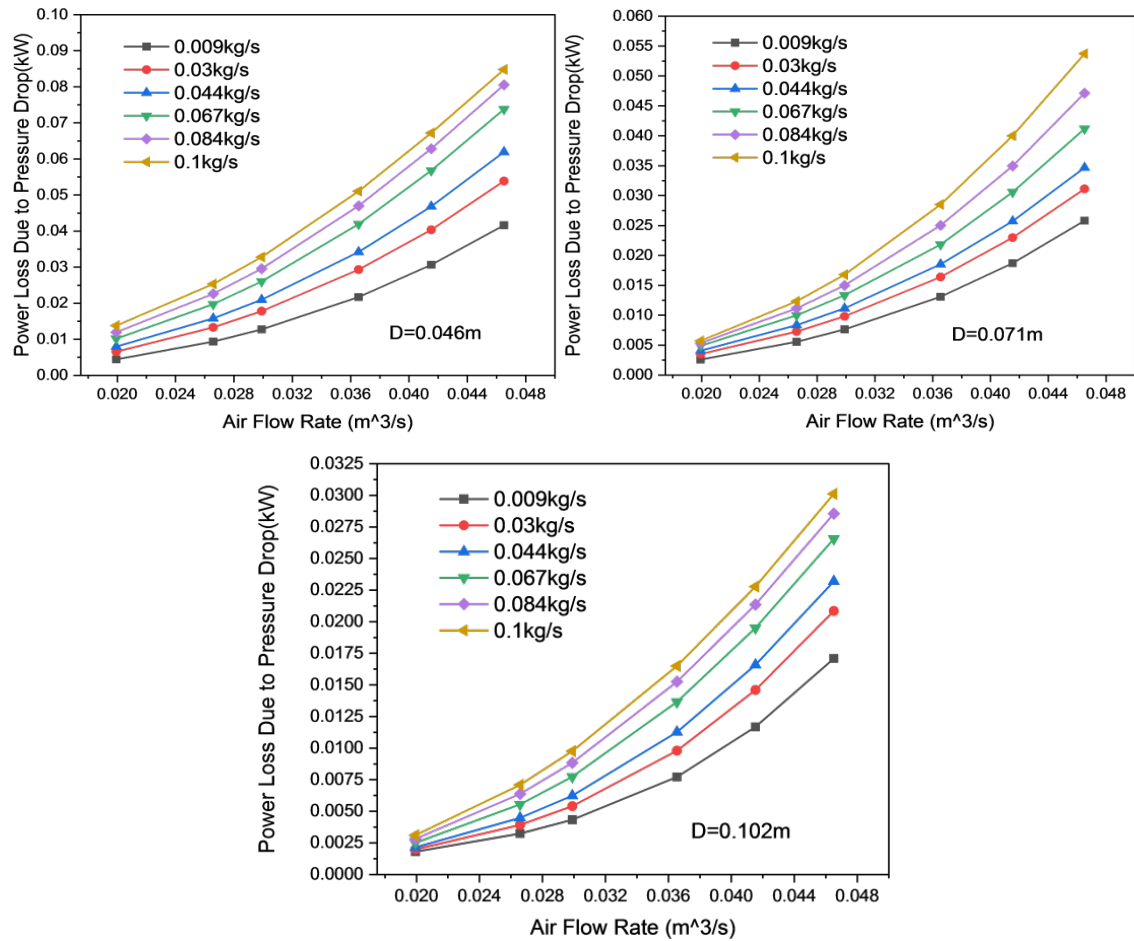


Figure 8 Power losses due to pressure drop

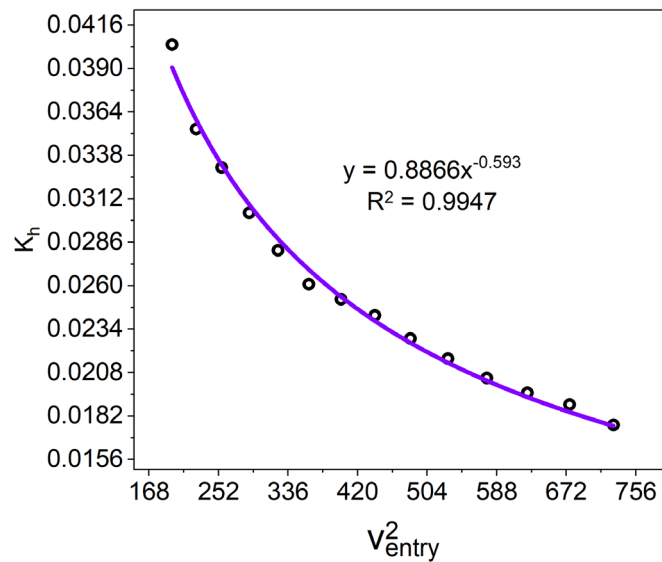


Figure 9 Pressure drop coefficient vs. inlet air velocity square ($12\text{--}28\text{ m s}^{-1}$)

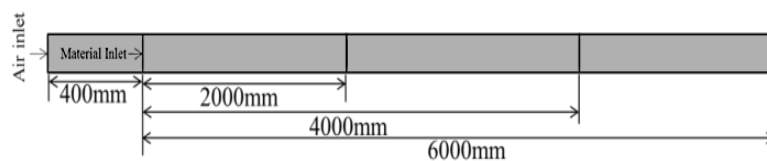


Figure 10 Geometry of simulation set up horizontal pneumatic teff grain conveyor

Figure 11 shows the pressure distribution for different conditions in the pneumatic conveying system. In panel (a), the pressure distribution for a pipe diameter of 102 mm, with an air velocity of 12 m s^{-1} , a pipe length of 2000 mm, and a mass flow rate of 0.009 kg s^{-1} , ranges from 0 to 46.85 Pa. In panel (b), the pressure distribution for a pipe diameter of 71 mm, with an air velocity of 28 m s^{-1} , a pipe length of 6000 mm, and a mass flow rate of 0.1 kg s^{-1} , ranges from 0 to 1287.88 Pa. Finally, in panel (c), the pressure distribution for a pipe diameter of 46 mm, with an air velocity of 28 m s^{-1} , a pipe length of 6000 mm, and a mass flow rate of 0.1 kg s^{-1} , ranges from 0 to 2472.11 Pa.

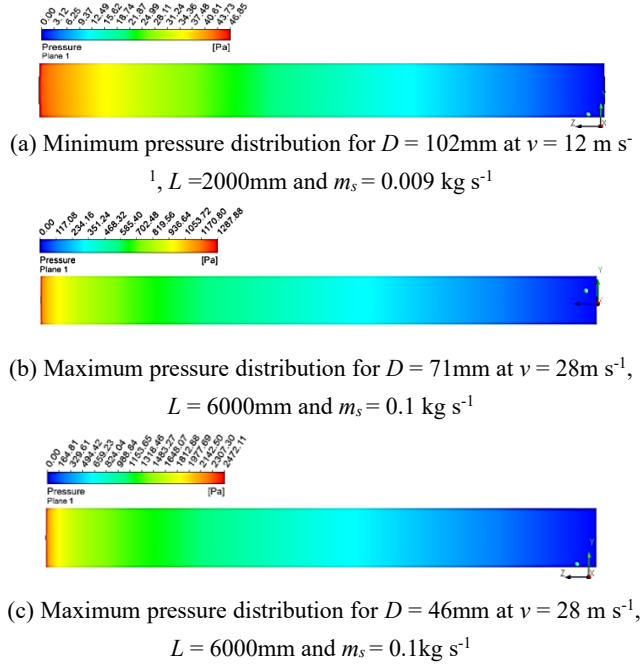


Figure 11 Pressure distribution

Figure 12 illustrates the air velocity distribution under different conditions in the pneumatic conveying system. In panel (a), the air velocity distribution for a pipe diameter of 102 mm, with an air velocity of 12 m s^{-1} , a pipe length of 2000 mm, and a mass flow rate of 0.009 kg s^{-1} , ranges from 0 to 13.61 m s^{-1} . In panel (b), the air velocity distribution for a pipe diameter of 71 mm, with an air velocity of 28 m s^{-1} , a pipe length of 6000 mm, and a mass flow rate of 0.1 kg s^{-1} , ranges from 0 to 34.37 m s^{-1} . Finally, in panel (c), the air velocity distribution for a pipe diameter of 46 mm, with an air velocity of 28 m s^{-1} , a pipe length of 6000 mm, and a mass flow rate of 0.1 kg s^{-1} , ranges from 0 to 35.67 m s^{-1} .

Figure 13 shows that for a pipe diameter of $D = 71 \text{ mm}$, inlet air velocity of $v = 20 \text{ m s}^{-1}$, material mass flow

rate of $m_s = 0.13 \text{ kg s}^{-1}$, and pipe length of $L = 4000 \text{ mm}$, the average value for the simulation is reached. The CFD-DPM simulation converged at iteration 150, with common scaled residuals for convergence ranging from $1e^{-3}$ to $1e^{-7}$. These residuals are typically used to indicate the accuracy and stability of the solution, with values below $1e^{-3}$ generally considered sufficient for convergence in most simulations.

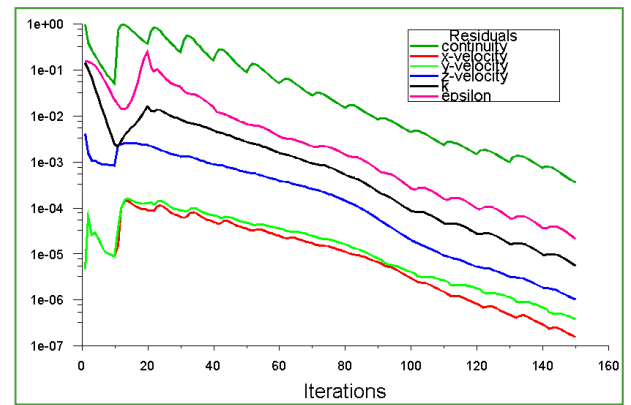
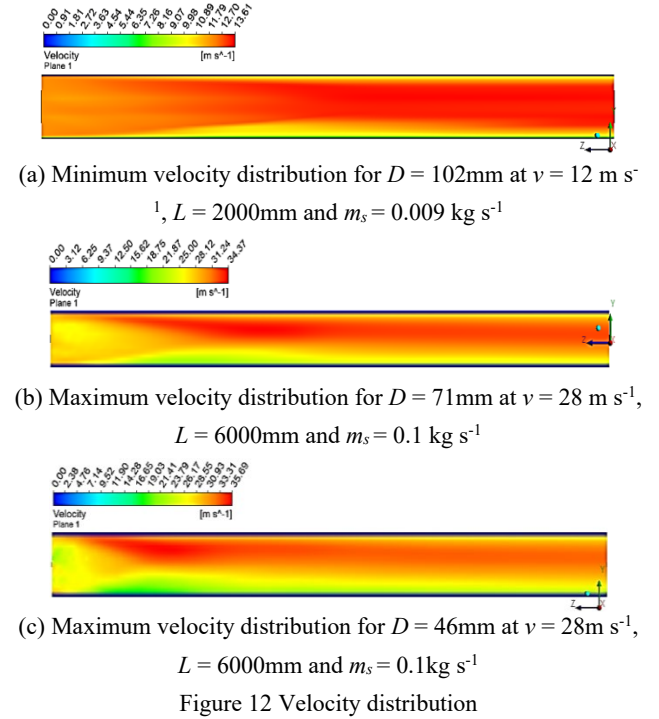
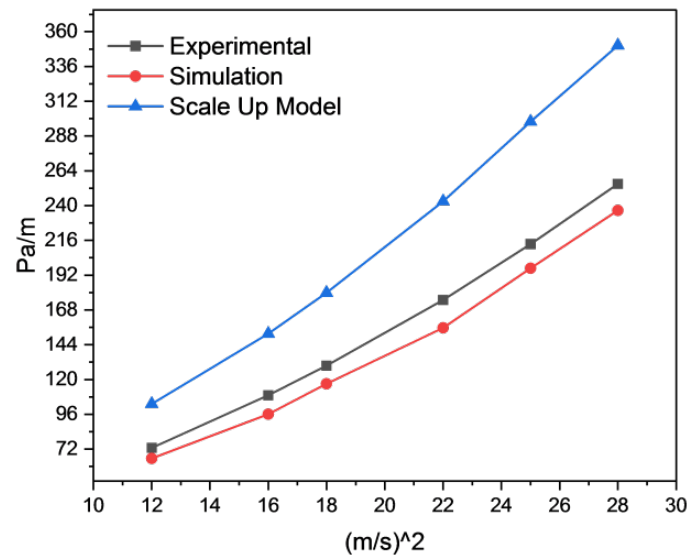


Figure 13 for pipe diameter $D = 71\text{mm}$, inlet air velocity $v = 20 \text{ m s}^{-1}$, material mass flow rate $m_s = 0.13\text{kg s}^{-1}$ and pipe length $L = 2000\text{mm}$

Figure 14 Findings compares experimental results with CFD-DPM simulation results, revealing strong agreement between the experimental and CFD-DPM results, with the simulation accurately reflecting experimental results the best curve fit established between airflow rate and power losses due to pressure drop.



(a) Pressure drop/unit length for experiment, simulation and scale up model

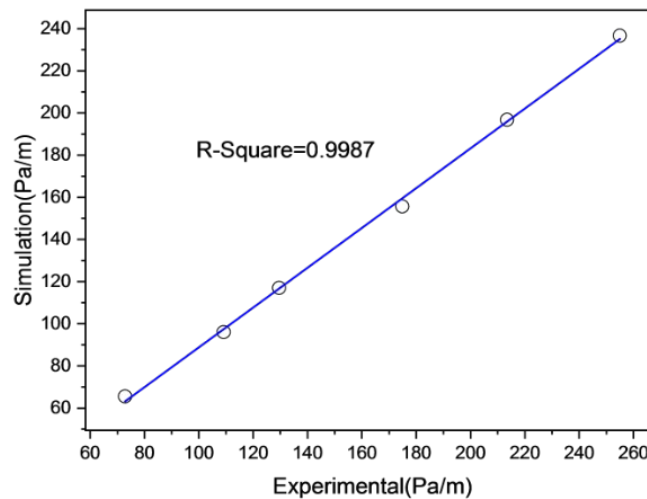
(b) Experimental (Pa m⁻¹) vs. simulation

Figure 14 Experimental and simulation

4 Conclusions

In a horizontal dilute phase pneumatic conveyor for teff grains, the pressure drop within the system increases with higher inlet air velocity and material flow rate, which is crucial for the design and operation of these systems? This relationship significantly influences energy requirements, operating costs, and overall efficiency. Understanding how inlet air velocity, material flow rate, and pressure drop interact is essential for optimizing performance and ensuring the reliable, cost-effective transport of teff grains through the conveyor.

As the pressure drop increases in a horizontal positive dilute phase pneumatic teff grain conveyor, the power loss due to the pressure drop also rises. This factor is critical in the design and operation of such pneumatic conveying

systems. The pressure drop is influenced by parameters like air velocity, grain flow rate, pipe diameter, and pipe length. Understanding the relationship between pressure drop and power loss is essential for optimizing energy efficiency and overall performance, ensuring reliable transport of teff grains.

The pressure drop coefficient in a horizontal dilute phase pneumatic teff grain conveyor has a strong correlation with the inlet air velocity square. This relationship is important for understanding energy requirements and optimizing pneumatic conveyance systems. The pressure drop coefficient is a crucial parameter that determines overall system performance, energy consumption, and operational costs for the pneumatic transport of teff grain.

Comparing experimental methods, simulation, and

scale-up models, CFD-DPM simulation proves to be more accurate than traditional scale-up models. When experimental analysis is costly or impractical, CFD-DPM simulations can effectively determine pressure drop in horizontal positive dilute phase pneumatic teff grain conveyors. This makes CFD-DPM a valuable tool for designing and optimizing pneumatic conveying systems for teff grain.

Acknowledgments

Adama Science and Technology University's heads of the Mechanical Engineering and Chemical and Food Engineering departments provided the laboratories needed for the study.

References

- ASAE. 1994. ASAE Standards S352.3: Moisture measurement unground grains and seeds. In *ASAE Standards 1994* (p. 469). ASAE, St. Joseph, MI.
- Asefa, B. G., D. F. Al Riza, R. B. Janius, S. Rosnah, and A. Shitu. 2023. Rapid classification of tef [*Eragrostis tef* (Zucc.) Trotter] grain varieties using digital images in combination with multivariate technique. *Smart Agricultural Technology*, 3: 100039.
- Cao, L., Q. Zhang, and R. Meng. 2022. CFD-DPM simulation study of the effect of powder layer thickness on the SLM spatter behavior. *Metals*, 12(11): 1561.
- Datta, B. K., and C. Ratnayaka. 2003. A simple technique for scaling up pneumatic conveying systems. *Particle and Particle Systems Characterization*, 21(3): 227–236.
- Dutta, P., and N. Nandi. 2019. Numerical study on turbulent separation reattachment flow in pipe bends with different small curvature ratio. *Journal of the Institution of Engineers (India): Series C*, 100(6): 995–1004.
- Dutta, P., S. K. Saha, N. Nandi, and N. Pal. 2016. Numerical study on flow separation in 90° pipe bend under high Reynolds number by k- ϵ modelling. *Engineering Science and Technology, an International Journal*, 19(2): 904–910.
- El-Behery, S. M., A. A. El-Haroun, and M. R. Abuhegazy. 2017. Prediction of pressure drop in vertical pneumatic conveyors. *Journal of Applied Fluid Mechanics*, 10(2): 527–535.
- Ghafori, H., and M. Sharifi. 2018. Numerical and experimental study of an innovative design of elbow in the pipeline of a pneumatic conveying system. *Powder Technology*, 331: 171–178.
- Huang, F., J. Zhang, N. Shrestha, Y. Cui, J. Zhao, Y. Zhu, M. Wang, and M. Yang. 2021. Role of CFD-based in silico modelling in establishing an in vitro-in vivo correlation of aerosol deposition in the respiratory tract. *Advanced Drug Delivery Reviews*, 170: 91–113.
- Jardon, E. L., A. Gallegos, J. A. Hernandez, and J. Siqueiros. 2019. Artificial neural networks (ANN) to predict overall heat transfer coefficient and pressure drop on a simulated heat exchanger. *International Journal of Applied Engineering Research*, 14(2): 156–187.
- Kaur, B., A. Mittal, P. Wypych, S. S. Mallick, and S. Jana. 2017. On developing improved modelling and scale-up procedures for pneumatic conveying of fine powders. *Powder Technology*, 305: 270–278.
- Kuang, S., K. Li, and A. Yu. 2020. CFD-DEM simulation of large-scale dilute-phase pneumatic conveying system. *Industrial & Engineering Chemistry Research*, 59(9): 4150–4160.
- Kuang, S., M. Zhou, and A. Yu. 2020. CFD-DEM modelling and simulation of pneumatic conveying: A review. *Powder Technology*, 365: 186–207.
- Ma, A. C., K. C. Williams, J. M. Zhou, and M. G. Jones. 2010. Numerical study on pressure prediction and its main influence factors in pneumatic conveyors. *Chemical Engineering Science*, 65(23): 6247–6258.
- Mezhericher, M., Vaclav Brosh, and Avi Levy. 2011. Modeling of particle pneumatic conveying using DEM and DPM methods. *Particle Science and Technology*, 29(2): 197–208.
- Özlü, R. R., and M. Güner. 2022. Determination of pneumatic conveying characteristics of canola seeds. *Tarım Bilimleri Dergisi*, 28(4): 656–665.
- Rahmanian-Koushkaki, H., S. H. Karparvarfard, M. H. Raoufat, H. Emdad, and K. Jafarpur. 2020. Estimating the mixture friction factor for the dense-phase transport of particulates in horizontal pipes with inner longitudinal spiral slots. *Biosystems Engineering*, 189: 51–61.
- Ratnayake, C., B. K. Datta, and M. C. Melaaen. 2007. A unified scaling-up technique for pneumatic conveying systems. *Particle Science and Technology*, 25(3): 289–302.
- Roberts, J., D. B. Hastie, and S. Zigan. 2022. Analysis and validation of a CFD-DPM method for simulating dust suppression sprays. *Particle Science and Technology*, 40(4): 415–426.
- Sharma, A., and S. S. Mallick. 2021. An investigation into pressure drop through bends in pneumatic conveying systems. *Particle Science and Technology*, 39(2): 180–191.

- Shijo, J. S., and N. Behera. 2021. Performance prediction of pneumatic conveying of powders using artificial neural network method. *Powder Technology*, 388: 149–157.
- Singh, J., H. S. Gill, and H. Vasudev. 2023. Computational fluid dynamics analysis on role of particulate shape and size in erosion of pipe bends. *International Journal on Interactive Design and Manufacturing*, 17(5): 2631–2646.
- Stone, L., D. Hastie, and S. Zigan. 2019. Using a coupled CFD-DPM approach to predict particle settling in a horizontal air stream. *Advanced Powder Technology*, 30(4): 869–878.
- Subramaniam, K., and W. S.-I. Wan Salim. 2024. A review of experimental approaches for investigating the aerodynamic performance of drones and multicopters. *Journal of Advanced Research in Experimental Fluid Mechanics and Heat Transfer*, 14(1): 1–24.
- Wang, C., W. Li, B. Li, Z. Jia, S. Jiao, and H. Ma. 2023. Study on the influence of different factors on pneumatic conveying in horizontal pipe. *Applied Sciences*, 13(9): 4881.
- Xu, Z., J. Zhou, H. Liu, J. Wang, and Y. Zhang. 2024. The numerical investigation of solid–liquid two-phase flow characteristics inside and outside a newly designed 3D sediment trap. *Journal of Marine Science and Engineering*, 12(1): 100.
- Yusop, F., S. Syazwan, N. N. Mat Hassan, K. A. Mohd Sari, M. A. A. Rahman, N. Hamidon, F. Ahmad, and M. K. Musa. 2023. Comparison of air flow distribution in the building for different position of air return. *Journal of Advanced Research in Numerical Heat Transfer*, 13(1): 81–86.
- Zhao, B., and Y. Su. 2010. Artificial neural network-based modeling of pressure drop coefficient for cyclone separators. *Chemical Engineering Research and Design*, 88(5–6): 606–613.
- Zhu, Q., H. Zhang, Z. Zhu, Y. Gao, and L. Chen. 2022. Structural design and simulation of pneumatic conveying line for a paddy side-deep fertilisation system. *Agriculture*, 12(6): 966.