EPJ Web of Conferences **340**, 08018 (2025)
Powders & Grains 2025 https://doi.org/10.1051/epjconf/202534008018

Computational Fluids Dynamics based Studies on Entrance length characteristics for High Concentration Particulate Coal Ash-Water Mixture Disposal

Ankit Prakash[1], and Anubhav Rawat[1]

[1]Applied Mechanics Department, Motilal Nehru National Institute of Technology Allahabad, Prayagraj, Uttar Pradesh, India -211004

Abstract. Pipeline transportation is a preferred mode of particulate material transportation due to its inherent advantages over other conventional modes of transportation, in particular, the High concentration solid fluid mixture system is developed to save huge specific energy consumption and less wear and tear to pipeline and attrition among particles at very high particle loading ratio. Although many works have been done on these systems, there is scarcity on understanding of the basic fluid mechanics. Therefore, the current work is focused on determination of entrance length for such system. Effect of various parameters like solid fluid mixture concentration, and Bingham Reynolds number are also studied. It is found that the entrance length for solid fluid mixture flows at high solid concentration substantially depends on the solid concentration, and flow regime both. Further, for turbulent flows the entrance length is independent of solid concentration. Entrance length for such flows are found to be more in case of laminar flows than turbulent flows.

1 Introduction

Thermal power plants are the lifeline of nations, particularly developing nations, where enormous cheaper electricity is required for the progressive growth of the country. Most of the existing thermal power plants are coal-based; which produce a massive amount of coal ash (fly ash and bottom ash) as waste. This needs to be disposed of safely, economically, and environment-friendly. One of the most affordable and sustainable modes of transportation is pipeline transportation of coal ash in the form of high-concentration solid liquid mixture (HCSD) which behaves as non-Newtonian fluid [1]. For any solid fluid transportation system, the pressure drop (measured in terms of pressure drop per unit length, i, pa/m) is referred as energy loss (amount of energy or force utilized to overcome the friction between the pipe's inner surface and liquid, and the viscous force acting between the fluid layers), which is majorly dependent on various geometrical and operational parameters, viz. pipe diameter (D), flow velocity (V), solid concentration (Cw), particle size of solid particulate material (d_{wm}), solid throughput (ST), etc. governed by flow physics ([1]-[6]). Out of these, flow velocity is a significant factor for pressure drop variation, as with the increase of flow velocity, pressure drop per unit length increases non-linearly[7] . One of those important aspects related to flow physics is the flow development length or the entrance length which is the region where flow becomes fully developed, and is characterized by constant centerline velocity, constant wall shear stresses and pressure drop per unit length of the pipe.

Mathematically, the fluid flow is considered to be developed when the radial velocity distribution, pressure distribution, and shear stress distribution remain constant, i.e., showing no further change along the flow direction. One of the most frequently referenced definitions of flow development length was articulated by Shah and London [8] for Newtonian fluids, for low Reynolds number flows is “the length of the fluid flow from entrance side required for the maximum cross-sectional velocity to reach 99% of its fully developed value for laminar flow”.

Some of the most common velocity profiles encountered in pipeline flows are the Uniform velocity profile; Parabolic velocity for laminar flow regime; and logarithmic and power law velocity profiles for turbulent flow regimes.

In uniform velocity profile, the velocity of the fluid particle does not change with respect to the position of the fluid normal to the pipe wall and is expressed as in **Fig. 1.** The present work focuses on the uniform inlet velocity profile only.

$$u(y) = U \tag{1}$$

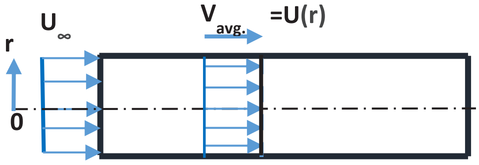


Fig. 1 Uniform Velocity profile

Since the fully developed flow is vital for any practical flow measurement application for their property determination, the entrance length plays a pivotal role in determining whether the flow is fully developed or still developing. Thus, it has been studied since the 18th century. Hagenback, Boussinesq, and Couette [9] are recognized as the earliest researchers who worked for entrance length determination. But the major breakthrough is given by Prandtl, who explained the boundary layer for viscous fluid. Later on, various other researchers studied the boundary layer effect and provided solutions for pipe flow in terms of Reynolds number [10].

Some other studies were conducted for non-Newtonian fluids especially Bingham fluids for the determination of entrance length [11] which states that the entrance region of Bingham plastic fluid flow has been studied extensively, with early analyses assuming a parabolic velocity profile, which was later deemed unrealistic. Improved methods incorporated mechanical balance equations for viscous dissipation to enhance accuracy. Later approaches determined velocity profiles using mass and momentum equations rather than assuming them. Numerical methods refined entrance flow analysis by discretizing continuity and momentum equations, revealing that boundary layer equations oversimplify entrance flow complexity due to assumed uniform pressure gradients. Systematic numerical studies showed that at high Reynolds numbers, these fluids deviate from Newtonian behavior, while at low Reynolds numbers, the correlation remains a weak and monotonic function of Bingham and Reynolds numbers.

The above literature survey reveals that, unlike Newtonian fluids, the studies of entrance length for Bingham plastic fluids have not been pursued more. Although an emphasis was given over the studies on pressure drop for Bingham fluids. Further, High concentration slurries (HCSD) which behave as Bingham plastic fluids have not been explored considering the parameters involved in their characterization for e.g., Particle size, Concentration, and the velocity or the associated Reynolds number is concerned.

Therefore, the current study emphasis on the development of a reliable correlation for entrance length for HCSD systems involving detailed parameters affecting it. The detailed computational fluid dynamics (CFD) study is conducted on straight circular pipe for establishing an accurate relation between entrance length and other parameters affecting the performance of HCSD mixture systems.

2 Computational Methodology

The présent computational study is conducted using the commercially available CFD software Ansys Fluent [12] to establish the non-dimensional entrance length and pressure drop correlation in terms of Reynolds number (Re), and solid concentration (C_w). The details of the computational methodology shall be explained in subsequent section.

2.1 Computational Domain and Meshing

A solid circular pipe is generated in the Ansys workbench Design Modular (**Fig. 2**). Based on the literature review done earlier, the dimensions of the current computational domain are selected in such a manner that the minimum length of the pipe remains more than 100D i.e., L>100D, where D is the diameter of the pipe. In case of laminar flow, the development length is expected to be more than that in case of turbulent flow. Therefore, two different lengths of the pipes are considered for the current study as shown in Table.1. To generate the structured mesh the computational domain is sliced into 5 parts which later combined to form a single geometry, as shown in **Fig.2.** A square prism is generated inside the cylinder, having the length of square side equal to a = 0.2D. The dimensions of the geometry so created are shown in the Table.1 below**.**

Table 1. Dimension of the computational domain

Parameter	Dimension
Diameter (D)	0.05m
Length (L)	6.0 m (Turbulent) 12.0 m (Laminar)
a	0.2 D

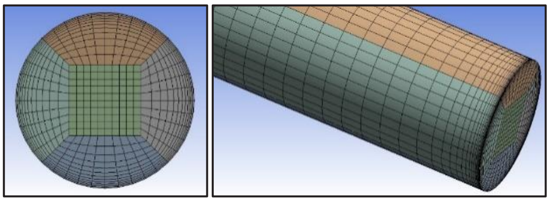


Fig. 2 Profile and cross-sectional view of computational domain and meshing

First order upwind scheme is considered for discretization, and under relaxation factors are set to default values. To discretize the computational domain the structured hexahedral cells are used. Further, inflation layers are also used at the boundary of the pipe wall to capture the wall effect. The inflation layer height is selected on the basis of Y+ equals to 1, with a growth rate of 1.2. Total 10 inflation layers are used for meshing. The meshed computational domain is as shown in **Fig. 2.**

2.2 Models, Solution Controls and Governing Equations

For viscous model in laminar regime, laminar model is selected and SST k-ω model is found to be most suitable capturing the physics of non-Newtonian HCSD Bingham fluid flow through pipes. In case of non-Newtonian Bingham Plastic fluid, the flow regime of the fluid is determined with the help of few non-dimensional numbers, i.e., Bingham Reynolds number (Re_B), critical Bingham Reynolds number (Re_{BC}), Hedstrom number (He) and dimensionless unsheared plug radius (x_c) which are evaluated with the help of Hanks Criteria [13].

In general, for the HCSD flow which behave as Bingham fluid, when Bingham Reynolds number is greater than critical Bingham Reynolds number the flow is assumed to be turbulent whereas when Bingham Reynolds number is less than critical Bingham Reynolds number then the flow is assumed to be laminar [13].

Further out of the various turbulence models mentioned in literature, SST k-ω model is found to be most suitable capturing the physics of non-Newtonian HCSD Bingham fluid flow through pipes [3-7, 9].

For current study i.e., steady state i.e., incompressible, with no source generation the governing equations are continuity and momentum equation (Eq. 2 and 3 below) respectively.

$$\nabla.(\rho\mathbf{u}) = 0 \qquad (2)$$

$$\nabla.(\rho\mathbf{uu}) = -\frac{\partial p}{\partial x} + \nabla.(\mu\,\nabla\mathbf{u}) \qquad (3)$$

and the above equations are solved using pressure-based solver.

2.3 Boundary Conditions and Fluid Properties

For the current computational study, velocity inlet is set as inlet boundary condition whereas, outflow is taken as outlet boundary condition. The walls of the pipe are considered as with no slip, having pipe surface roughness constant as 0.5 and average roughness height is calculated using water flow data available in literature [14].

The high concentration coal ash particle liquid mixture behaves as homogeneous Bingham Plastic fluid and their rheological equation is given as below.

$$\tau = \tau_o + \mu\dot{\gamma} \qquad (4)$$

Where, τ is shear stress, τ_o is yield shear stress, μ is solid fluid mixture viscosity and $\dot{\gamma}$ is shear strain rate or strain rate.

The rheological properties like yield shear stress, Bingham viscosity, density for the present computational study are given in the Table.2 below. The weighted mean particle size, d_{wm}=34.5μm and specific gravity of fly ash is taken as 2.06 [2-3].

Table 2. Rheological properties of fly ash

C_w (%)	ρ_m (kg/m^3)	τ_o (Pa)	μ (Pa. s)
55	1394.72	0.173	0.007
60	1446.63	0.317	0.0136
62	1468.49	0.484	0.0282
65	1502.63	0.736	0.0502
68	1538.23	1.673	0.1388

2.4 Grid Independence Test (GIT)

In current study, GIT is performed by obtaining the maximum centreline velocity at 60% conc. (by wt.) and flow velocity of 1m/s across pipe diameter at 0.5 m and 6 m from the inlet of the pipe. A grid with mesh elements of 11.5 x 10^5 is selected. Some of the selected meshes and their variations are represented in tabular form in Table 3 below.

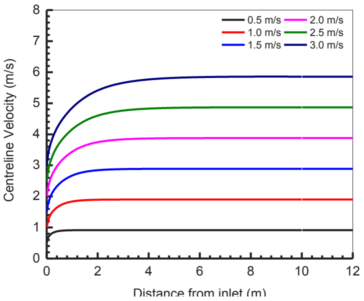
Table 3. Rheological properties of fly ash

Mesh Elements	Velocity (at 0.5 m from inlet)	% Deviation (at 0.5m from inlet)
286488	1.22	
646461	1.24	1.64
1152576	1.25	0.8

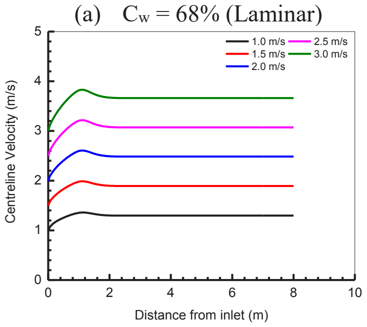
3 Results and Discussion

In case of uniform velocity profile at inlet, the velocity of fluid at any point with respect to position remains same (Eq.1). The determination of entrance length for uniform velocity profile is given by Ookawara et al. [15]. For laminar flow, entrance length is said to be achieved where the centreline velocity reaches 99% of its fully developed value. Whereas, for turbulent flows it is said to be achieved on the first appearance of fully developed value of centreline velocity after the peak value of centreline velocity.

To ascertain that a graph between the centreline velocity and the distance from inlet of the pipe are drawn in **Fig. 3.** for laminar and turbulent regimes.



(a) C_w = 68% (Laminar)



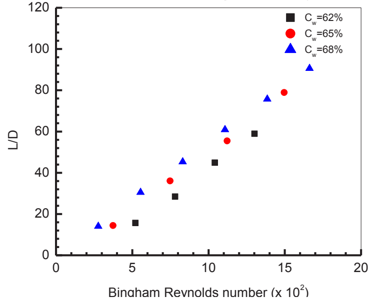
(b) C_w = 60% (Turbulent)

Fig. 3 Centerline Velocity at various positions from inlet for uniform velocity profile

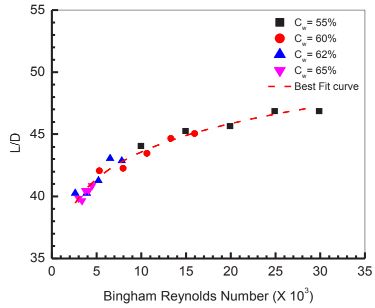
Fig. 3. depicts the nature of centreline velocity with respect to position from inlet for laminar and turbulent flow regime. Unlike turbulent flow regime there is no peak velocity in case of laminar flow regime as the flow centreline velocity starts from given input velocity and increases non-linearly until it achieves the maximum centreline velocity and after that the flow becomes constant with respect to position. Thus, the distance from inlet where flow velocity becomes constant is known as entrance length and after that, the flow is known as the developed flow.

Form **Fig.3** the distances L at which the centreline velocities become constant are calculated and a graph between non-dimensional entrance length (L/D) (where, L is entrance length, and D is pipe diameter) and Bingham Reynolds number is plotted at various concentration (C_W), and velocity (V) for both turbulent and laminar flow regimes as shown in **Fig. 4. Fig.4** clearly depicts that the change in L/D ratio with the change in Bingham Reynolds number is significant in case of Laminar flow and as the concentration and Bingham Reynolds number increases the non-dimensional entrance length also increases **(Fig.4 (a))**. Whereas, for turbulent flows the Bingham Reynolds numbers also show significant effect over the L/D but the effect of concentration is insignificant on L/D

(**Fig.4 (b)**). These results are explainable based on the basic physics of laminar and turbulent flows which state that the laminar flows exhibit viscous virtues whereas, turbulence flows are generally inertial flows.



(a) Laminar Flow



(b) Turbulent Flow

Fig. 4 Non-Dimensional Length Vs Bingham Reynolds Number (Re_B) at various C_w

To obtain a generalised mathematical function between L/D and Re_B for both laminar and turbulent flow, a power law allometric model is fitted in form of Eqs. 5 & 6 respectively.

$$\frac{L}{D} = (0.15C_w^{2.6})Re_B^{9.875\times10^{-1}} \tag{5}$$

$$\frac{L}{D} = 22(Re_B)^{7.5\times10^{-2}} \tag{6}$$

The above Eq.5 is valid for Bingham Reynolds number in the range of 250 < Re_B < 2,100 (laminar flow), whereas Eq.6 is valid for Bingham Reynolds number in the range of 2,500 < Re_B < 30,000 (turbulent flow).

The modelled relation between L/D, Re_B, and C_w shows good agreement with experimental data for laminar and turbulent flow, respectively.

The relations established in Eqs. 5 & 6 shows good approximation having deviation of ± 20% for laminar flows and ±5% for turbulent flows. The average deviation for laminar and turbulent flows are 2% and 0.7% respectively.

4 Conclusion

The present study primarily focuses on the effect of Uniform velocity profile on the entrance length for High Concertation Particulate Coal Ash and water slurry flows. From the current study, following conclusions may be drawn:

The entrance length is the major parameter affecting the amount of pressure energy required for HCSD systems, which is dependent significantly on the type of on the velocity profile.

It is found that the non-dimensional entrance length (L/D) is maximum for Uniform velocity profile in laminar flows, whereas minimum for uniform velocity profile, in turbulent flows.

The non-dimensional entrance length increases with respect to Bingham Reynolds number.

Non-dimensional entrance length is independent from solid concentration in turbulent flow regime.

References

1. A. K. Verma, S. N. Singh, V. Seshadri, Effect of Particle Size Distribution on Rheological Properties of Fly Ash Slurries at High Concentrations. International Journal of Fluid Mechanics Research 33 (5): 445–57. (2006). https://doi.org/10.1615/INTERJFLUIDMECHRES.V33.I5.50.
2. Chandel, Sunil, S. N. Singh, V. Seshadri, Transportation of High Concentration Coal Ash Slurries through Pipelines. International Archive of Applied Science and Technology,1(1) (2010). https://soeagra.com/iaast/iaastjune2010.htm.
3. A. Rawat, S. N. Singh, V. Seshadri, Computational Methodology for Determination of Head Loss in Both Laminar and Turbulent Regimes for the Flow of High Concentration Coal Ash Slurries through Pipeline." Particulate Science and Technology 34 (3): 289–300 (2016). https://doi.org/10.1080/02726351.2015.1075637.
4. A. Rawat, S. N. Singh, V. Seshadri, Erosion Wear Studies on High Concentration Fly Ash Slurries. Wear 378–379 (May):114–25 (2017). https://doi.org/10.1016/J.WEAR.2017.02.039.
5. A. Rawat, S. N. Singh, V. Seshadri, Variation of Physical and Rheological Properties of Fly Ash Slurries with Particle Size and Its Effect on Hydraulic Transportation at High Concentrations." Particulate Science and Technology 37 (2): 151–60 (2019). https://doi.org/10.1080/02726351.2017.1352636.
6. A. Rawat, S. N. Singh, V. Seshadri, CFD Analysis of the Performance of Elbow-Meter with High Concentration Coal Ash Slurries, Flow Measurement and Instrumentation 72 (April):101724 (2020) https://doi.org/10.1016/J.FLOWMEASINST.2020.101724.
7. A. Prakash, A. K. Upadhyay, A. Rawat, Optimization of Specific Energy Consumption in the High Concentration Slurry Disposal Pipelines for Coal Ash in Thermal Power Plants, International Journal of Coal Preparation and Utilization, February (2024). https://doi.org/10.1080/19392699.2024.2321362.
8. R.K. Shah, A.L. London, Circular Duct, Laminar Flow Forced Convection in Ducts, January, 78–152 (1978) https://doi.org/10.1016/B978-0-12-020051-1.50010-3.
9. P.J. Pritchard, J.W. Mitchell, *Fox and McDonald's introduction to fluid mechanics*. John Wiley & Sons (2016).
10. J.R. Bodoia, J. F. Osterle, Finite Difference Analysis of Plane Poiseuille and Couette Flow Developments, Applied Scientific Research 10 (1): 265–76 (1961) https://doi.org/10.1007/BF00411919/METRICS.
11. R.J. Poole, R. P. Chhabra, Development Length Requirements for Fully Developed Laminar Pipe Flow of Yield Stress Fluids, Journal of Fluids Engineering, Transactions of the ASME 132 (3): 0345011–14 (2010) https://doi.org/10.1115/1.4001079/456696.
12. ANSYS Fluent User's Guide, Release 16.2., ANSYS, Inc. ANSYS, Inc, Canonsburg. (2016)
13. R.W. Hanks, On the Flow of Bingham Plastic Slurries in Pipes and Between Parallel Plates, Society of Petroleum Engineers Journal 7 (04): 342–46. (1967) https://doi.org/10.2118/1682-PA.
14. V. Singh, S. Kumar, S. K. Mohapatra, Modeling of Erosion Wear of Sand Water Slurry Flow through Pipe Bend Using CFD, Journal of Applied Fluid Mechanics 12 (3): 679–87 (2019). https://doi.org/10.29252/JAFM.12.03.29199.
15. S. Ookawara, K. Ogawa, Flow properties of Newtonian and non-Newtonian fluid downstream of stenosis, Journal of chemical engineering of Japan, *33*(4), pp.582-590. (2000)