Multidimensional Analysis of the Transport Characteristics of Clay Balls in a Digital Slurry Pipeline

Sheng Li[1] ,Yu Zhou[2] , Zenghai Yang[1] , Sheng Wang[2*]

[1]CCCC Shanghai Dredging Co., Ltd., Shanghai, China
[2]National Engineering Research Center of Dredging Technology and Equipment, Shanghai, China

*Corresponding author: 1249931984@qq.com

Abstract. During the dredging process, the characteristics of slurry transport in the pipeline are closely related to dredging efficiency. In certain transport conditions, the slurry in the pipeline may carry clay ball particles, which can affect transport efficiency. This paper uses a CFD-DEM model and a two-way coupling method to conduct numerical simulation analysis of the transport characteristics of clay ball particles in a slurry pipeline. After model validation using field detection data, the movement state and flow field distribution of clay balls in the slurry were analyzed. The following conclusions were drawn: After individual particles and particle groups enter the pipeline, their velocity decreases rapidly, especially for particle groups, which quickly settle to the bottom of the pipe and accumulate. The front particles move faster, while the rear particles move more slowly and gradually stop, eventually moving slowly to the pipe outlet under the influence of the slurry. The pressure in the pipeline decreases significantly along the way, and a local low-pressure area appears around the clay balls. The presence of clay balls occupies part of the flow cross-section, causing the flow velocity in that region to increase, while the lowest velocity occurs around the clay balls.

Keywords: numerical simulation; transport characteristics; homogeneous slurry; clay ball

1 Introduction

With the continuous increase in modern engineering demands, slurry pipeline transport technology has become increasingly widespread in dredging projects, slurry transportation, construction, and other fields. Dredging involves underwater excavation of soil and rock to clear, widen, or deepen rivers, lakes, and other bodies of water using mechanical methods [1]. It is widely applied in excavating new waterways, ports, and canals, deepening and cleaning existing channels and ports, clearing rivers and reservoirs, as well as land reclamation and removing underwater obstacles [2]. In dredging projects, dredged materials such as clay, sand, and gravel are primarily transported through pressurized pipelines to designated locations. Pipeline transport requires a certain amount

Y. Zhang et al. (eds.), *Proceedings of the 2024 10th International Conference on Architectural, Civil and Hydraulic Engineering (ICACHE 2024)*, Advances in Engineering Research 259,
https://doi.org/10.2991/978-94-6463-658-1_87

of pressure, but due to the resistance within the dredging pipeline, the slurry velocity gradually decreases, and when it falls below the critical velocity, there is a risk of pipeline blockage[3].Pipeline transport has become a crucial method for long-distance transportation of particulate materials due to its advantages of being environmentally friendly, cost-effective, and having good continuity[4]. However, the movement behavior of particles within the pipeline and their interaction with the fluid are very complex. This is especially important when the particle size and shape vary significantly, as they greatly affect the transport characteristics such as pipeline pressure and velocity distribution. In recent years, numerical simulation has emerged as a powerful research tool, widely used to analyze the movement characteristics of particles in slurry and their effects on the flow field[5-6]. Slurry transport in dredging falls under the category of solid-liquid two-phase flow, and current research methods in this field include experimental studies, theoretical analysis, and numerical simulations. Numerical simulations allow for a detailed understanding of the movement state of particles with different sizes, their accumulation process, and the influence they have on the slurry flow field within the pipeline[7]. This is of great practical significance for optimizing pipeline transport system design, improving transport efficiency, and reducing energy consumption.

2 Numerical Model

2.1 Geometric Model Construction

Based on a specific section of the pipeline in the dredging project site, a three-dimensional model of a horizontal pipeline with an inner diameter D=850mm and a length of L= 60D is established. The pipeline is meshed using an O-type grid. In the fluid region inside the pipeline, a hexahedral mesh is used, while the boundary layer is divided into 15 layers, with a growth rate of 1.2 for each successive layer. The total number of mesh elements is approximately 1.27 million. The cross-sectional view of the meshed pipeline in the computational model is shown in Figure 1.

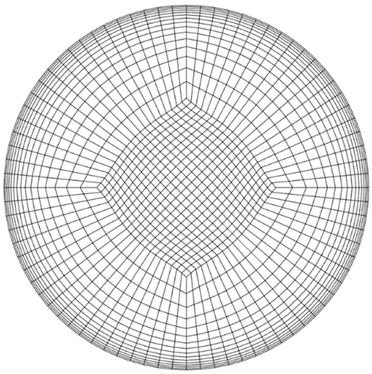

Fig. 1. Cross-sectional View of the Pipeline Mesh

2.2 Boundary Conditions

Boundary conditions are one of the critical steps in numerically solving fluid flow problems. They describe the behavior of the fluid at the boundaries of the computational domain, ensuring that the numerical simulation accurately reflects the physical situation. The following conditions are applied in this study: Velocity Inlet: The velocity at the inlet is specified, along with the volume fractions of the liquid and solid phases, which are set based on different operational conditions. This ensures that both fluid and particles enter the pipeline with defined properties.Pressure Outlet: The outlet pressure is set to approximately 13.0 kPa, based on the results from field pressure sensors.

Pipe Wall Properties: The pipeline is assumed to be a welded steel pipe with slight corrosion. The absolute roughness of the pipe wall is specified as 0.1 mm. The flow shear condition between the slurry and the pipeline wall is modeled using the no-slip condition, meaning that the slurry's velocity at the wall is zero relative to the wall. The gravitational acceleration in the vertical direction (Z-axis) is set to -9.81 m/s^2, representing the effects of gravity on the slurry transport.Numerical Methods: The governing equations are discretized using a first-order upwind scheme, and the solution method employs a phase-coupled algorithm to account for the interactions between the liquid and solid phases. The simulation is performed using transient analysis, and the convergence criterion for the residuals is set to 10^{-3}. Figure 2 shows the schematic diagram of the boundary conditions for the CFD model.

Fig. 2. Schematic Diagram of Boundary Conditions for the CFD Model

3 Numerical Results Analysis

3.1 Single Particle Calculation Results

Figure 3 shows the movement of a clay ball particle with a diameter of 0.1 meters within the pipeline. It can be observed that upon entering the pipeline, the velocity of the clay ball rapidly decreases to approximately 2 m/s within 1 second, followed by a sharp drop. Within 2 seconds, the clay ball comes into contact with the bottom of the pipeline, and its velocity reduces to around 1.5 m/s. Subsequently, it is slowly carried toward the pipeline exit by the slurry, with its speed remaining roughly at 1 m/s during the process, while gradually decreasing.

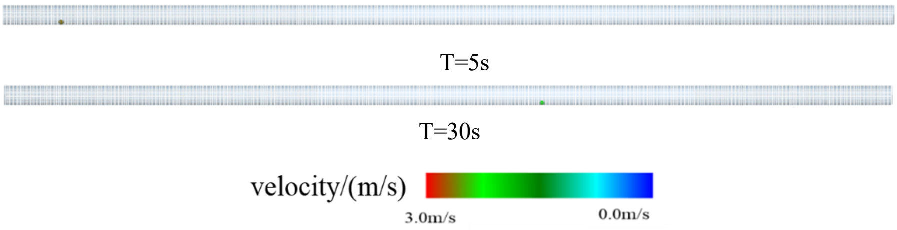

Fig. 3. Motion of Clay Ball Inside the Pipeline

Figure 4 presents the pressure and velocity distribution of the slurry inside the pipeline at T = 20s. It can be seen that the presence of the single clay ball has a minimal impact on the overall pressure distribution within the pipeline. The pressure exhibits a gradual attenuation along the length of the pipeline, with lower pressure observed near the pipe walls at the downstream end. The calculated pipeline frictional resistance is approximately 0.38 kPa/m. The velocity variation within the pipeline is also relatively small, with only slight velocity changes occurring near the clay ball, where a minor flow recirculation can be observed around the particle.

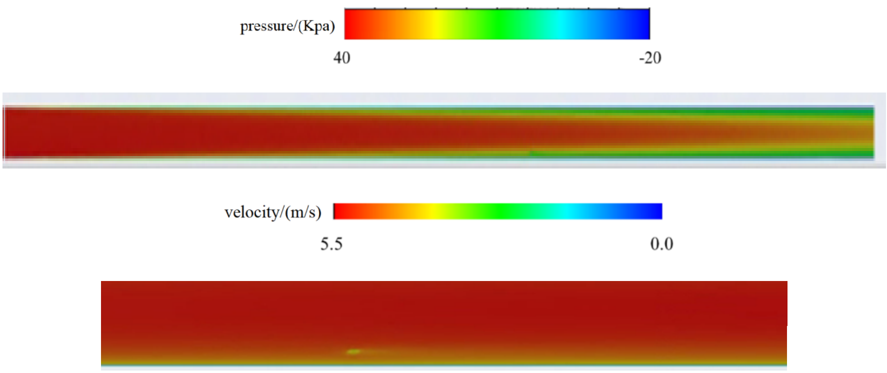

Fig. 4. Pressure and Velocity Distribution Diagram Inside the Pipeline

3.2 Clay Ball Group Calculation Results

Figure 5 shows the distribution of clay balls at different time intervals. It can be observed that after the clay balls enter the pipeline with the initial flow velocity, their speed quickly decreases, and they rapidly settle to the bottom of the pipeline. Initially, the clay balls accumulate at the bottom of the pipeline, where collisions between particles occur. Eventually, the clay balls flatten out along the bottom of the pipe. The clay balls at the front maintain a relatively higher velocity, while those at the rear slow down almost to a stop. The distance between the clay balls increases over time. Under the influence of the slurry, the clay balls slowly move forward, gradually advancing toward the pipeline outlet.

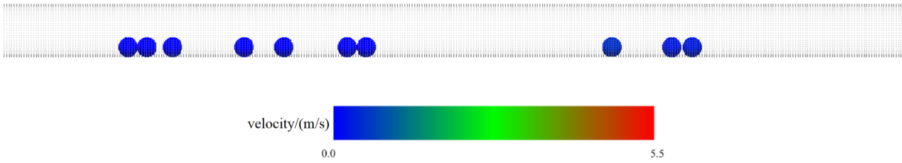

Fig. 5. Clay particle distribution at the time of T=30 seconds.

Figure 6 presents the velocity and pressure distribution of the slurry inside the pipeline at T = 30s seconds. It can be observed that the pressure decreases along the length of the pipeline. Near the clay balls, localized low-pressure areas appear, and in the downstream section of the pipeline, low-pressure regions are distributed near the pipe walls due to the lower velocity in those areas. The calculated frictional resistance along the pipeline is approximately 0.48 kPa/m.

The velocity distribution shows an increase in flow velocity at the cross-sections where the clay balls are present, as the clay balls occupy part of the flow cross-section. Since the flow rate remains constant, the velocity must increase to compensate. The lowest velocity areas are found around the clay balls, as the large clay particles obstruct the slurry's movement.

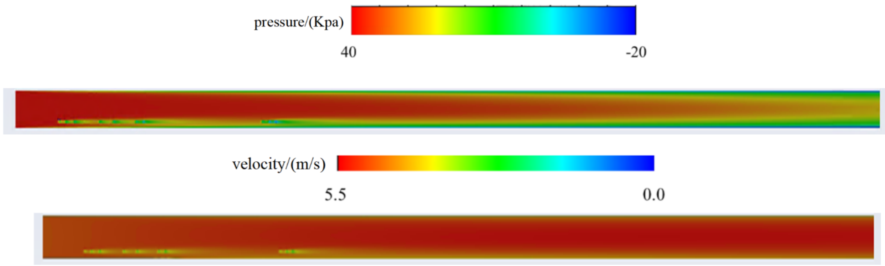

Fig. 6. Shows the pressure and velocity distribution within the pipe.

4 Conclusion

This paper introduces a numerical method for simulating the transportation of clay spheres in slurry pipelines. The study reveals that when single particles or particle clusters enter the pipeline, their velocity decreases rapidly, especially for particle clusters, which quickly settle at the bottom of the pipeline and accumulate. The front-end particles have higher velocities, while the rear particles gradually decelerate until they come to a stop. Eventually, they are slowly transported to the pipeline outlet under the flow of the slurry. Meanwhile, the presence of clay spheres significantly disturbs the flow field within the pipeline, causing noticeable pressure attenuation along the pipeline and forming localized low-pressure zones around the clay spheres. As the clay spheres occupy part of the cross-sectional flow area, the flow velocity in these regions increases, while the minimum velocity is observed around the clay spheres. These findings are of great significance for optimizing the design and operation of slurry pipeline transportation systems.

References

1. Cai Y . (2013) Study on structure optimization and characteristic parameters of air-aided conveying device dredging ship. Wuhan University of Technology.
2. Lin Y, Yuan BL and Zhao ZH (2019). "Technology of dredging pipeline's air-injected transportation in Maluan Bay, Xiamen," Port and Waterway Engineering, (z2):74-77.).
3. Tang B, Kong JY and Rao RS (2008). "The experimental research on assisting transportation by ejecting compressed-air to pipeline," Journal of Hubei University of Technology, 23(03):81-83+86.
4. Tang B, Kong JY and Rap RS (2011). "Experimental research on air-ejecting device for pipeline air transportation," Machinery Design and Manufacture, (6):152-154.
5. Xiong T, Fan SD, Zhu HH and Yang ZX (2011a). "Study on resistance characteristic by air injection for slurry in dredger pipelines," Journal of Wuhan University of Technology (Transportation Science and Engineering), 35(3):554-557.
6. Xiong T (2011b). "Study on aeration transportation technology of cutter suction dredger pipeline," Wuhan University of Technology.
7. Yu LW (2006a). "Research on experiment of three-phase flow pipeline transportation and numeric simulative analysis," Wuhan University of Technology.