Research Calculation Study of a Slurry Circulation System based on the CFD-DEM Method

Yuhua Wang, Liying Deng

School of Chemical Equipment, Shenyang University of Technology, Liaoyang 111000, China

Abstract

The slurry circulation system is a key link in the construction of slurry balance shield to ensure the stability of tunneling and the smooth transportation of muck. Its design parameters and flow patterns are directly related to construction safety and efficiency. Based on a certain intercity rail transit project in Guangzhou, this paper theoretically calculates the key parameters of the slurry circulation system during the construction process and conducts numerical simulation using the CFD-DEM method. The results show that the pressure loss along the straight pipe section and the local resistance of the curved pipe can be obtained through simulation, and the total system head calculated based on this is basically consistent with the pump layout on site. Further flow field analysis indicated that there was a large velocity gradient near the wall of the straight pipe, while secondary flow and local vortices occurred inside the elbow; There are obvious stratified movement characteristics during the particle transport process, and the wall area of the elbow is prone to accumulation and frequent collisions, which may induce local wear and even blockage.

Keywords

Slurry Circulation System; CFD-DEM; Pressure Loss; Pump Arrangement; Particle Transport.

1. Introduction

The slurry balance shield is widely used in various projects[1] such as river underpasses, undersea tunnels and urban rail transit due to its strong adaptability. Among them, the slurry circulation system is the core link in the construction of slurry shield, and its operational efficiency directly affects the stability of tunneling, the capacity of muck transportation and the balance[2].However, in actual engineering, the slurry often contains particles of different sizes such as sand, pebbles and crushed rocks, which cause the flow in the pipeline to present strong turbulence and non-Newtonian characteristics, posing a great challenge to the accurate calculation of pressure loss. In response to this problem, scholars at home and abroad usually develop equations based on physical model tests, numerical simulations, or a combination of both to approximate the relationship between pressure loss and related influencing factors. Wasp et al. proposed an iterative method for predicting the total pressure loss caused by the slurry based on the fact that the total pressure loss in a two-phase flow can be divided into the pressure loss caused by uniformly dispersed particles and the additional pressure loss[3]caused by the formation of the slurry layer. Kaushal and Tomita conducted experiments on micron-sized particle water suspensions and improved the Wasp model by reducing the impact of some restrictive assumptions on the pressure drop and proposed an empirical pressure drop equation[4]. Miedema proposed the pressure drop equation[5] for a mixture of sand-water and gravel-water with millimeter-sized particles. Qin et al.[6] systematically summarized methods for calculating pressure loss in the slurry shield circulation system based on theoretical derivation. With the development of numerical simulation, Yang et al. used the CFD-DEM method to reveal the exponential growth of pressure drop with velocity and volume fraction, and pointed out the significant influence[7] of inclination angle on pressure drop. Yidong Guo et al. combined the H-B rheological model with the CFD-DEM coupling calculation to establish a new model of slurry transport pressure gradient, and verified its effectiveness[8].

Traditional theoretical formulas have certain deviations when dealing with complex flows of granular slurry, while the CFD-DEM method can simultaneously consider the non-Newtonian rheological properties of the slurry and the particle motion, and evaluate pipeline pressure drop, pump head requirement and system stability more accurately. Therefore, this study takes a section of an intercity railway line in Guangzhou as the engineering background, calculates the key parameters of the slurry circulation system based on the actual operating conditions, and uses a CFD-DEM coupling approach to establish straight-pipe and bend-pipe models in order to investigate the flow behavior and pressure-loss characteristics of non-Newtonian slurry in the pipeline. On this basis, the total head of the system is determined, suitable pumps and pump arrangement are designed, and the flow-field characteristics and particle transport behavior are further analyzed.

2. Calculation of Key Parameters of the Slurry Circulation System

2.1 Project Overview

This study is based on an intercity rail transit project in Guangzhou that adopts the slurry shield tunneling method. The geological conditions along the tunnel alignment are complex, with an extensive overburden and diverse stratigraphic units. As can be seen from the geological profile in Fig 1, the buried section of the tunnel is covered with mixed fill soil, plain fill soil and loose layers such as fine sand and medium-coarse sand, and there are strata such as pebbles and hard plastic clay in some areas. The underlying bedrock consists mainly of sandy mudstone and argillaceous siltstone, with zones of strong, moderate, and slight weathering locally developed. Overall, it is characterized by an alternating distribution of the upper loose overburden and the lower hard bedrock.

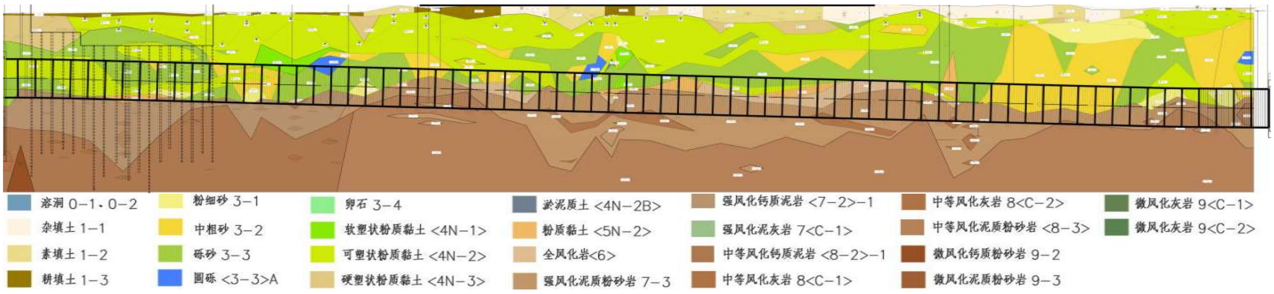

Fig. 1 Geological profile of the left line of a certain engineering section

2.2 Calculation of Key Parameters of the Slurry Circulation System

The operation of the slurry circulation system depends on the reasonable matching of key parameters, especially during shield propulsion, the slurry must simultaneously have good fluidity,sufficient support pressure and effective carrying capacity of the muck.[9] Therefore, in this section, the key parameters of the circulation system, such as flow rate, velocity, volume concentration and pipe diameter, are calculated and verified in combination with the actual working conditions of the project, providing basic data for the system layout and subsequent numerical simulation.

The amount of debris produced by the shield machine during the tunneling process mainly depends on the excavation diameter and the tunneling speed. The muck discharge can be calculated from the following equation:

$$Q_0 = \frac{60\pi {D_C}^2 V_S}{4\times 1000} \tag{1}$$

In the formula, D_C is the excavation diameter, taken as 8.84m; V_S is the tunneling speed, with a maximum of 60mm/min. At this point, the amount of slag Q calculated is 220.84m^3/h. In the actual tunneling process, V_S=20mm/min is taken, and Q_0=54.83m^3/h.

In addition, considering the influence of soil moisture content on the excavation volume, the maximum amount of excavated soil and sand can be further calculated:

$$G=(1-k)\times\frac{60\pi {D_C}^2 V_S}{4\times1000} \tag{2}$$

Where k is the moisture content, take 20%; When V_S is 60mm/min, G=177m^3/h.

According to the law of conservation of mass, the inflow and outflow of slurry should satisfy the following formula:

$$Q_2=Q_1+Q_0 \tag{3}$$

$$\rho_2 Q_2=\rho_1 Q_1+\rho_0 Q_0 \tag{4}$$

Further, from Equations 1 and 2, the volume flow rate of the inlet pipe is derived as:

$$Q_1=Q_0\times\frac{\rho_0-\rho_2}{\rho_2-\rho_1} \tag{5}$$

Where ρ_0 is the density of undisturbed soil (2680kg/m^3), ρ_1 is the density of the incoming slurry (1150kg/m^3),and ρ_2 is the discharge slurry density (1310kg/m^3); When the shield tunneling speed reaches a maximum of 60mm/min, the feed flow rate Q_1 is calculated to be 1890m^3/h; Substitute Equation 1 and calculate Q_2=2111m^3/h.

The pipe velocity is then obtained from the flow rate as follows:

$$v^{'}=\frac{4\times Q}{\pi\times D^2} \tag{6}$$

Substituting the data yields v_1=5.46m/s, v_2=6.1m/s, which are the theoretical requirements of the system for flow rate and velocity at the maximum tunneling speed. In the actual design, the conventional propulsion speed can be used as the calculation condition, or the minimum safe velocity can be determined in combination with the critical suspension flow rate criterion to ensure the suspension of particles while taking into account energy consumption and wear control.

Based on the flow rate and velocity of the slurry, the commonly used empirical formula[10] for calculating the pipe diameter is:

$$D\geq 4.6\sqrt{\frac{Q}{V}} \tag{7}$$

Where Q is the flow rate; V is the flow velocity. The flow velocity of the slurry in the metal pipe is 2 to 5m/s. Take 5m/s. Substitute the data to calculate $D_1 \geq 317$mm, $D_2 \geq 340$m. To simplify procurement and on-site management, adopt the arrangement of the same inlet and outlet pipe diameters, and in combination with engineering experience, select a pipe diameter of 350mm.

The critical sedimentation flow rate is the lowest flow rate at which solid particles in a pipeline remain suspended and do not settle. To ensure smooth transportation, the design flow rate of the slurry must be higher than this critical value. The formula for calculating the critical sedimentation flow rate can use the Wasp calculation formula[3]:

$$V_c = F_L \left[2gD_2 \left(\frac{\gamma_s - \gamma_2}{\gamma_2} \right) \right]^{\frac{1}{2}} \left(\frac{d_2}{D_2} \right)^{\frac{1}{6}} \tag{8}$$

Where FL is the concentration of solid particles and the particle size constant, take 1.34;and d2 is the particle size of the slag, 20mm; Substitute the data and calculate that the critical flow velocity of the slurry discharge pipe is 2.2m/s.

To ensure smooth muck transport and maintain particles in suspension or motion, it is common practice to adopt a safety factor above the critical value; accordingly, the minimum design velocity v2 for the discharge pipe must satisfy:

$$v_2 = 1.2 \sim 1.25 V_c \tag{9}$$

Substitute the data and calculate v_2=2.8m/s, with the flow rate within the reasonable range of 2 to 5m/s commonly used in engineering. This velocity is used as an inlet boundary condition in the subsequent CFD-DEM simulations to verify that particles can be effectively carried and transported to the surface. Although this velocity satisfies the critical suspension requirement, practical design should also consider slurry rheology, particle-size distribution, and pipeline configuration to further reduce the risks of sedimentation and clogging. According to the pipe flow formula, the volume flow rate of the slurry discharge pipe is:

$$Q_2 = \frac{\pi \cdot {D_2}^2}{4} \cdot v \tag{10}$$

In the formula, D_2 is the diameter of the pipe; Substitute the data and calculate Q_2=970m^3/h; According to Equation 1, Q_1=748m^3/h ; v_1=2.2m/s is calculated.

In summary, a pipe diameter of 350 mm is adopted. The critical-velocity check indicates that the minimum design velocity should exceed 2.8m/s to prevent particle deposition. These parameters provide baseline inputs for the system arrangement and serve as reliable boundary conditions for the subsequent CFD-DEM simulations.

3. Numerical Methodology

3.1 Numerical Mode

The CFD-DEM coupling method combines computational fluid dynamics (CFD) with the discrete element method (DEM) for simulating fluid-solid interactions. The fluid flow is described by the locally averaged Navier-Stokes equations, solved using the traditional CFD method[11], and the discrete element method applies Newton's laws of motion to each particle to simulate solid-phase motion. In this numerical calculation, FLUENT was used to simulate the fluid phase, and EDEM was used to simulate the discrete slag rock. The CFD-DEM coupled simulation model flow of the slurry pipe slag discharge is shown in Fig 2.

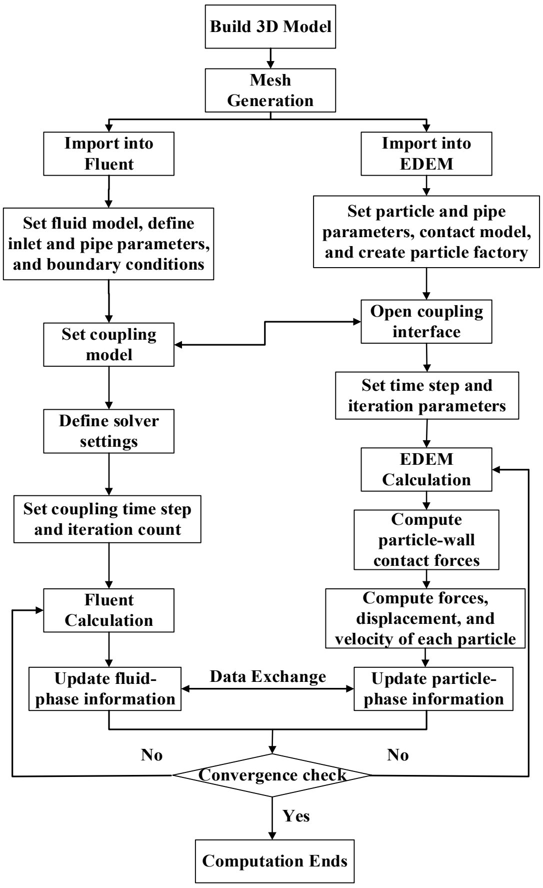

Fig. 2 Flowchart of the simulation calculation

3.1.1 Fundamental Theory of Fluid Phases

This study does not consider heat transfer within the slurry, and the slurry flow process follows the law of conservation of mass and the law of conservation of momentum.Continuity equation:

$$\frac{\partial \alpha_f \rho_f}{\partial \mathrm{t}} + \nabla \cdot \left(\alpha_f \rho_f U_f\right) = 0 \tag{11}$$

Momentum equation (Navier-Stokes equation) :

$$\frac{\partial}{\partial \mathrm{t}}\left(\alpha_f \rho_f U_f\right) + \nabla \cdot \left(\alpha_f \rho_f U_f\right) = -\nabla \rho_f + \nabla \alpha_f \tau + \alpha_f \rho_f g + F_{pf} \tag{12}$$

3.1.2 Fundamental Theory of Particle Discrete Elements

The debris in the slurry is a granular phase and is subject to the forces of other debris, pipe walls and slurry, and its movement and collision can be processed by Newton's second law using the discrete element method.

Translational equation:

$$m_i \frac{dU_{\mathrm{p},i}}{dt} = m_i g + \sum_{j=1}^{n} \left(F_{c,ij} + F_{\mathrm{d},ij} \right) + \left(F_{c,iw} + F_{\mathrm{d},iw} \right) + F_{f,i} \tag{13}$$

Rotation equation:

$$I_i \frac{d\omega_i}{dt} = \sum_{j=1}^{n} T_{ij} + T_{iw} \tag{14}$$

3.1.3 Interaction between Fluid and Particles

The slurry and particles are affected by buoyancy FB, drag force FD and pressure gradient force FP, and the calculation formulas are as follows:

$$\mathbf{F}_B = -\frac{\rho_f}{\rho_p} \mathbf{g} \tag{15}$$

$$\mathbf{F}_D = \frac{18\mu_e}{\rho_P d_p^2} \frac{C_D Re_{\mathrm{p}}}{24} \left(\mathbf{U}_f - \mathbf{U}_p \right) \tag{16}$$

$$\mathbf{F}_P = -\frac{1}{\rho_p} \nabla p_f \tag{17}$$

3.2 Geometric Models and Meshing

As shown in Fig 3, in this paper, the three-dimensional geometric models of the straight pipe and the curved pipe section were established through Solidworks. The pipe diameter was uniformly set at 350mm, and the length of the straight pipe was taken as 5m to ensure that the flow field was fully developed before the outlet. Smooth arc transitions are used at the bends, and the curved sections include common engineering arrangements such as horizontal, vertical, and inclined.

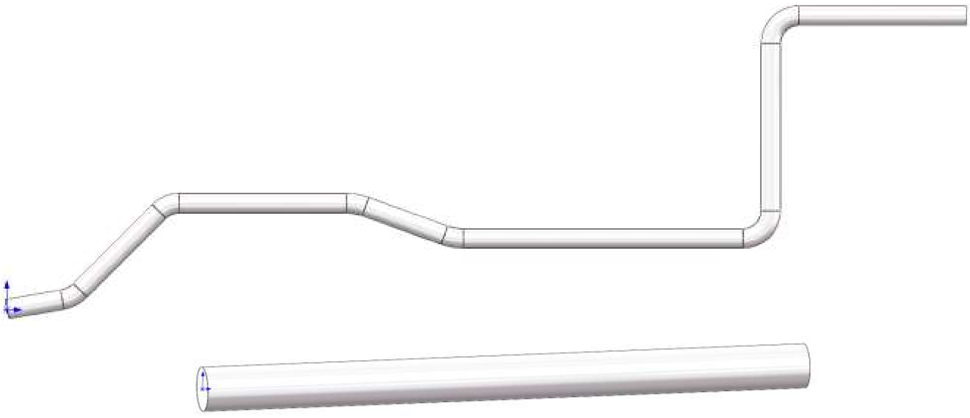

Fig. 3 Geometric model of the fluid domain of the slurry pipe

As shown in Fig 4, meshing is carried out using Workbench Mesh, and the meshing is generated using a multi-region method based on geometric structure, considering the requirements for mesh quality in fluid-structure coupling calculations. The surface mesh is optimally partitioned and boundary layer dilation is set to improve the accuracy of the near-wall area calculation.

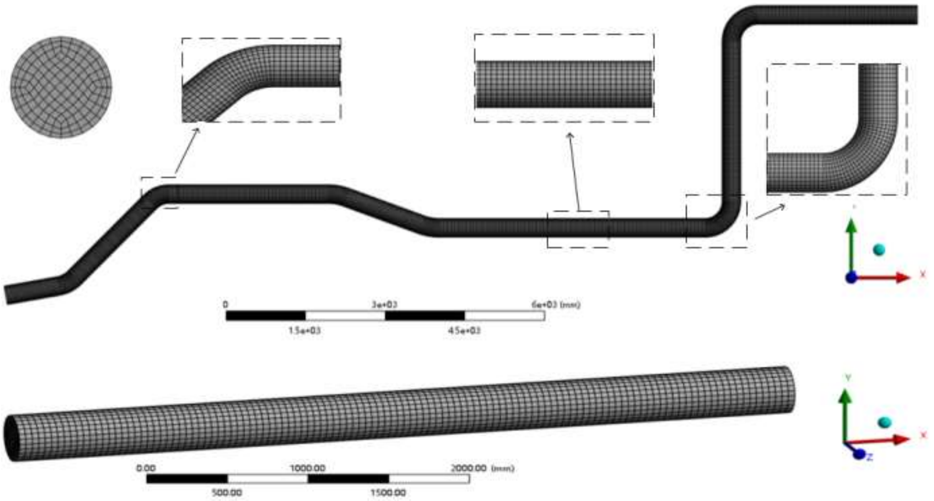

Fig. 4 Meshing of the fluid domain

3.3 Boundary Condition Settings

In the CFD model, set velocity inlet, pressure outlet, pipe inner wall without slip boundary, gravity direction set the same as in the EDEM software, Y=-9.8m/s^2 , using the standard k-ε turbulence model, Herschel-Bulkley type non-Newtonian fluid model. In the DEM model, a particle factory is established at a distance from the inlet to continuously produce particles, with a propulsion velocity of 20mm/min, and the corresponding muck production rate is 54.83 kg/s. Other parameters are set as shown in Table 1[12].

Table 1. Parameter Settings in CFD-DEM Coupled Simulation

Parameters	Value
Fluid density(kg/m^3)	1150
Yield stress of fluid(Pa)	4.598
Plastic viscosity of fluid(Pa·sn)	0.886
Power law index of fluid	0.654
Particle diameter(mm)	20
Particle density(kg/m^3)	2680
Poisson ratio of particle	0.25
Shear modulus of particle(Pa)	1e+08
Pipe wall density(kg/m^3)	7800
Poisson ratio of pipe wall	0.25
Shear modulus of pipe wall(Pa)	1e+08
Particle-particle static friction coefficient	0.35
Particle-particle rolling friction coefficient[7]	0.01
Particle-pipe wall static friction coefficient[13]	0.35
Particle-pipe wall rolling friction coefficient	0.01
Coefficient of restitution	0.05

Select the Eulerian-Lagrangian model[14] through the coupling interface. The simulation ignores the smaller forces acting on the particles and focuses on the effects of gravity, drag, and lift on the gravel. Set up gispow drag force models, Saffman lift and pressure gradient forces, etc. in the coupling module. In the Fluent and EDEM coupling process, this paper sets the Fluent calculation time step to 50 times[15]. Set the Fluent time step to 0.0005s and the EDEM time step to 0.00001s.

3.4 Validation of Coupling Model Validity

To verify the effectiveness of the CFD-DEM model in pressure loss prediction in this study, the solid-liquid two-phase pipeline transport experiment established by Vlasak et al.[16] was selected as a reference. Based on the experimental setup and measurement conditions, the corresponding CFD-DEM numerical model was constructed to simulate some pipe sections, and the obtained pressure loss was compared with the experimental results. The comparison results are shown in Fig 5. It can be seen that as the flow rate of the mixture increases, both the simulated values and the experimental values show an upward trend with the increase of flow rate; The deviation between the two was controlled within the range of 1% to 12%, which was within the allowable precision range for engineering. This indicates that the CFD-DEM coupling method can reasonably reflect the transport characteristics of the fluid-solid two-phase in the pipeline and can be used for predicting the pressure drop along the way during slurry transportation and numerical simulation of the sludge discharge system.

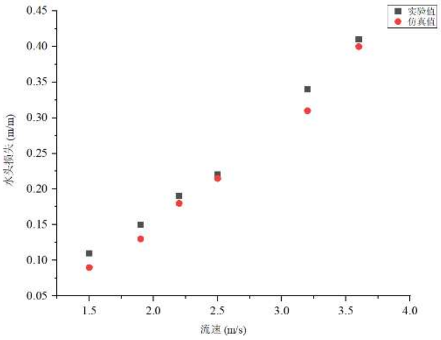

Fig. 5 Comparison of pressure loss simulation and experiment

4. Calculation Results and Analysis

4.1 Pipeline Pressure Loss Calculation and Pump Placement

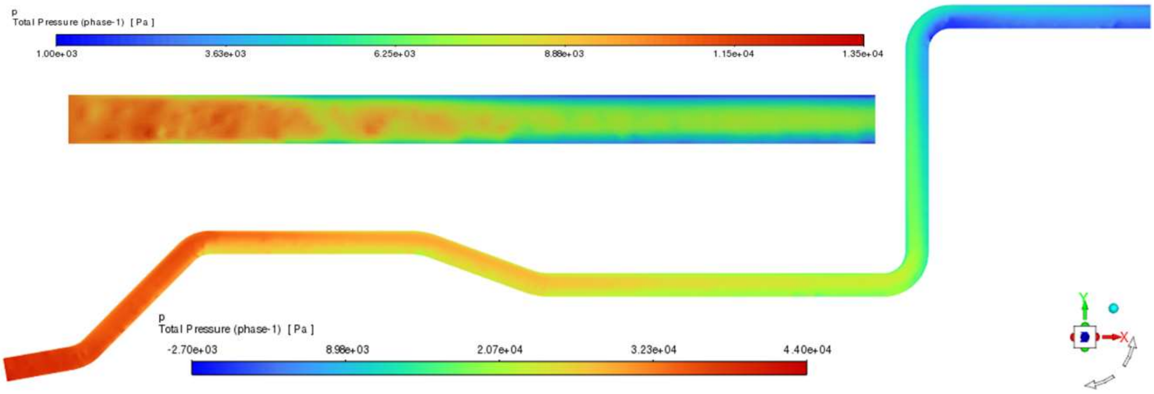

Fig. 6 Schematic diagram of pressure distribution in straight pipe sections and curved pipe sections

As shown in Fig 6, the pressure loss results of the straight and curved sections were obtained through CFD-DEM numerical simulation. After the pipeline is stably transported, the calculation shows that the pressure difference between the inlet and outlet of the 5-meter-long straight pipe is approximately

4800Pa, then the pressure loss ΔP along the horizontal straight pipe is 960Pa/m; The pressure difference between the inlet and outlet of the elbow section is approximately 35,000 Pa.

Formula for calculating head loss coefficient per unit length:

$$h_f = \times \frac{\Delta P}{\rho_2 \cdot g} \tag{18}$$

In the formula, ρ_2 is the density of solid-liquid mixture in the pipe, 1310kg/m^3 . Substituting the data results in a simulated head loss coefficient of 0.074 per meter, which is higher than the theoretical formula calculation because the slurry, as a non-Newtonian fluid, has a higher apparent viscosity and a significantly increased frictional factor; At the same time, large particle size generates additional drag force and collision friction during transportation, resulting in additional momentum dissipation and an overall increase in the pressure drop level. Substituting the simulation data of the curved pipe, the sum of the head loss along the curved pipe and the head loss at the elbow is 2.7m.

The formula for calculating the required head of the slurry pump is:

$$He = H_f + H_\omega + H_{\mathrm{pd}} + H_{tb} - \frac{p_{\min}}{\rho_2 \mathrm{g}} \tag{19}$$

$$H_f = h_f \times Le \tag{20}$$

$$Le_2 = L + H_{\mathrm{pd}} + L_2 + H_{\mathrm{tb}} \tag{21}$$

In the formula, H_{pd} represents the depth of the shaft, taken as 20 meters. H_{tb} is the height of the sludge-water separation equipment, 10m; $p_{\min}$ is the minimum working pressure, 0.5×10^5Pa; L_2 is the distance from the shaft to the sludge-water separation equipment, taken as 300m; L is the total length of the tunnel. Here, subtract the length of the curved part and take 1146 meters. Substitute the data and calculate H_e as 139 meters. The WARMAN 12/10G-GH slurry pump, with a maximum head of 77m, was selected based on the slurry flow rate and head. It was placed behind the shield as the main pump. Additionally, two WARMAN 12/10G-GH slurry pumps were chosen as relay pumps and shaft lift pumps.

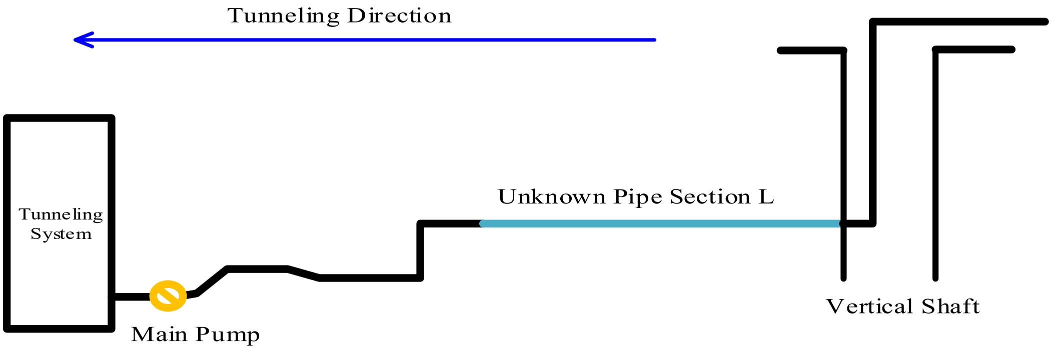

Fig. 7 Schematic diagram of the slurry pipe pump layout

As shown in Fig 7, take the outlet pressure of the main pump of the slurry pipe as approximately 7.86bar, or 0.786MPa. Assuming L is the distance of the unknown horizontal section, to find the position of the relay pump, the following formula holds:

$$\Delta P = \Delta P_a \cdot L + \Delta P_\mathrm{b} \tag{22}$$

In the formula: ΔP_a is the loss per unit length along the horizontal straight pipe calculated by the fluid-solid two-phase model; ΔP_b is the total pressure loss of the elbow section. Substituting the data, the distance L of the unknown horizontal section is calculated to be 782.3m. That is, the slurry pump is set up 782m from the unknown horizontal section. According to the actual layout of the slurry pumps on site, the position of the relay pump is 750m. This is because it is necessary to ensure that the slurry pressure in the pipeline is always maintained within a safe range to avoid the pressure being too low and causing poor transportation or even affecting the construction progress. From this, it can be seen that the calculation results of pressure loss in the slurry discharge pipeline based on CFD-DEM coupling can provide a reliable reference for determining the position of the relay pump and have strong guiding value for the layout of the pumping station on site.

4.2 Analysis of Flow Field Characteristics

4.2.1 Slurry Velocity Distribution

As shown in Fig 8, the velocity distribution of the slurry in the slurry discharge pipe shows distinct stratified characteristics. The velocity of the slurry gradually decreases from the center of the pipe to the wall, and there are also differences in velocity changes at different cross-sectional positions. When the slurry flows through the elbow, the flow structure changes significantly due to the combined effects of inertia, centrifugal force and pressure gradient. The main flow of the slurry migrates after colliding on the inner side of the elbow, resulting in a significantly higher flow velocity on the inner side than on the outer side (1-1 and 3-3 sections). At the same time, the flow separation and increased turbulence intensity formed a secondary flow perpendicular to the main flow direction,which pushed the slag-rock particles towards the outer wall, gradually shifting the high-speed flow layer to the outer side (2-2 and 4-4 sections), intensifying the particle collision and concentration aggregation in the area, causing higher energy loss and local frictional pressure drop, resulting in greater overall resistance in the elbow section than in the straight section.

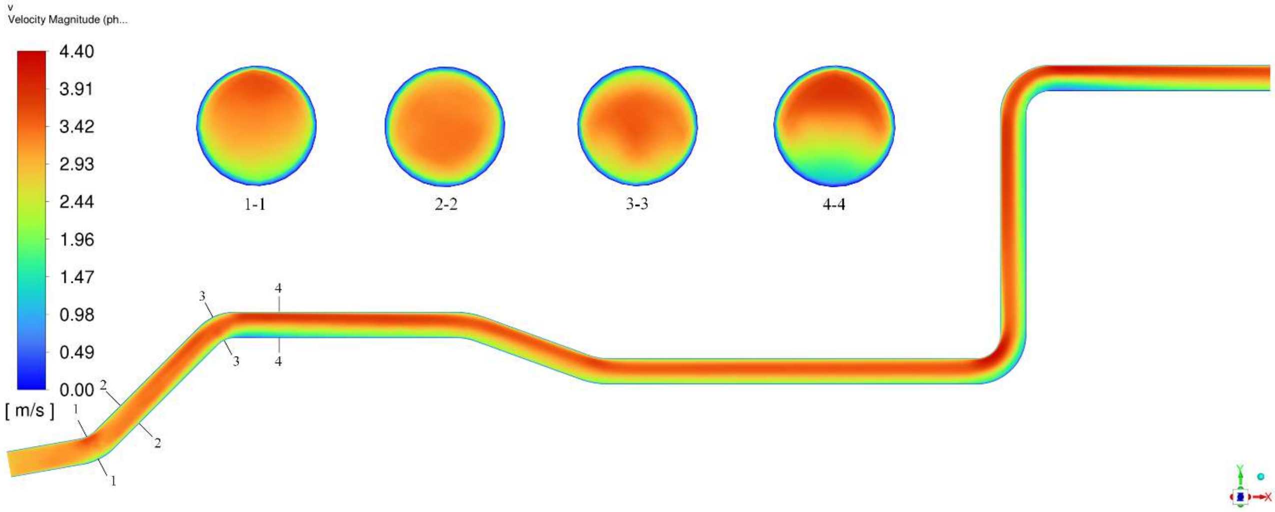

Fig. 8 Cloud diagram of the liquid phase velocity distribution

4.2.2 Slurry Pressure Distribution

As shown in Fig 9 is the cloud map of the pressure distribution of the slurry pipeline in a stable conveying state. The pressure inside the pipeline decreases gradually from the inlet to the outlet as a whole. In straight sections, the pressure distribution is relatively uniform and decreases steadily, mainly due to fluid viscosity friction loss and particle acceleration energy loss. In the elbow area, due to the combined effect of inertial force and centrifugal force on the fluid, the flow field shifts, and the collision and turbulent disturbance between the fluid and the inner wall of the elbow intensify energy dissipation, creating a local high-pressure loss area. The pressure distribution is non-uniform, and the pressure on the outer arc side is higher than that on the inner arc side, forming a pressure gradient.

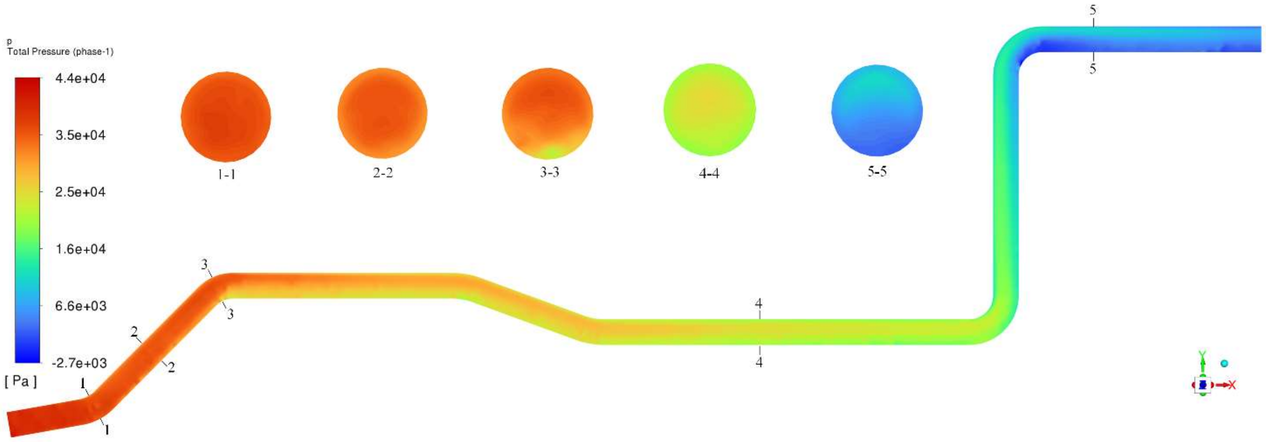

Fig. 9 Cloud diagram of the liquid phase pressure distribution

4.3 Particle Property Analysis

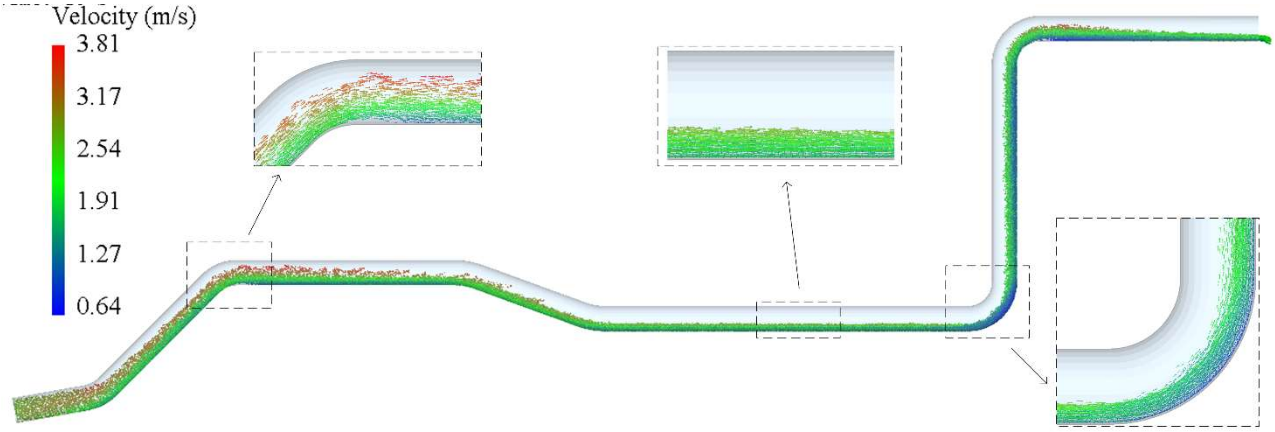

Fig. 10 Particle phase velocity distribution diagram

As shown in Fig 10, the particle velocity distribution cloud diagram is presented in the stable conveying state of the slurry pipe. When the particles enter the pipe and are pushed along the bottom of the pipe, the particle movement velocity shows a clear decreasing trend from top to bottom. The higher movement speed of the particles in the upper part of the pipe indicates that there is less accumulation of particles in this area and they are more affected by fluid dragging. The closer to the bottom of the pipe, the slower the particle velocity decreases, indicating an increase in particleaccumulation and greater resistance to movement, which is consistent with the fluid velocity distribution and the influence of gravitational settling. From the partial enlarged vector diagram of particle motion in the elbow section in Fig , it can be seen that when the particles enter the horizontal pipe from the inclined pipe, the slag and stone particles move in a “ bundle ” with the fluid as a whole at the elbow, and then, due to inertia, their trajectories are parabolic forward. Some particles collide with the bottom of the horizontal pipe, resulting in velocity attenuation and easy deposition at the bottom of the pipe. The rest of the particles continue to move towards the outlet under the drag of the fluid.

5. Conclusion

(1)The slurry flow rate and the minimum flow velocity in the pipe were determined through condition analysis, and it was verified that they could meet the basic requirements for normal slurry transportation.

(2)The straight and curved pipe numerical models were established to obtain the pressure drop characteristics along the way and in the local area. Based on the numerical results, the head of the pumping station was calculated and the layout scheme was designed. The calculation results were basically consistent with the actual layout on site, indicating that the method based on CFD-DEM numerical simulation can provide an effective reference for pump type selection and the position of the relay pump.

(3)The analysis of the flow field characteristics indicates that the slurry flow in the straight pipe section is generally uniform, and the velocity gradient in the near-wall area is obvious, which is the main area for energy dissipation. The curved section shows typical secondary flow and vortex structures, resulting in concentrated local energy loss in the outer curved wall area.

(4)The particle transport analysis shows that under the set conditions, most of the particles can be carried forward by the fluid, achieving stable transport. But there is still sparse deposition at the bottom of the straight pipe and particle aggregation and frequent impacts on the outer wall of the curved pipe, which are potential wear and clogging risk areas. Therefore, in practical engineering, it is necessary to maintain a sufficient flow rate to prevent deposition and take wear-resistant measures at key locations.

References

[1] National Innovation Plan for Urban Rail Transit in China: Dev elopment Background, General Requirements and Development Goals [J]. Urban rail transit,2025,(08):29-35.

[2] Wei Xiangming. Key Technology Research on Mud Treatment for the Slurry Circulation System of the Shield in Whole Clay Strata [J]. Railway Constru ction Technology,2018,(07):77-80.

[3] Wasp E J:Solid-Liquid flow slurry pipeline transportation.(Trans Tech Publications, Switzerland 1977).

[4] Kaushal D,Tomita Y .Solids concentration profiles and pressure drop in pipeline flow of multisized particulate slurries [J]. International Journal of Multiphase Flow, 2002,28(10): 1697-1717.

[5] Miedema A S .A head loss model for homogeneous slurry transport for medium sized particles[J].Journal of Hydrology and Hydromechanics,2015,63(1):1-12.

[6] Qin Bangjiang, Xia Yimin. The Differential Pressure Analysis and Calculation of Pipeline Circulation System in Slurry Shield [J]. Chinese Hydraulics Pneumatics and Seals, 2016,36(07):52-55.

[7] Yang D ,Xia Y ,Wu D , et al.Numerical investigation of pipeline transport characteristics of slurry shield under gravel stratum[J].Tunnelling and Underground Space Technology incorporating Trenchless Technology Research,2018,71:223-230.

[8] Yidong G ,Xinggao L ,Dalong J , et al.A new pressure gradient model of slurry shield pipeline system coupling with rheological properties and wall slip behaviour under sandy stratum[J].Tunnelling and Underground Space Technology incorporating Trenchless Technology Research,2023,134.

[9] Wei Shunhua, Zhang Hua, Qiu Jian. Design and Selection of Mud Circulation System for Mud Balance Shield Machine[J]. Modern Manufacturing Technology and Equipment, 2022,58(08):73-78.

[10] Chen Wenyuan. Research on Design and Calculation of Slurry Circulation System for Slurry Balance Shield [J]. China Equipment Engine ering,2018,(02):86-89.

[11] Kuang Shibo. Compuational study of pneumatic conveying by diserete element model [D]. Northeastern University,2009.

[12] Yidong G ,Xinggao L ,Yu S , et al.Investigation into the flow characteristics of slurry shield pipeline system under sandy pebble stratum: Model test and CFD-DEM simulation[J].Powder Technology,2023, 415.

[13] Shengqiang J ,Xiaodong C ,Guodong C , et al.Optimization of fresh concrete pumping pressure loss with CFD-DEM approach[J].Construction and Building Materials,2021,276.

[14] Muhammad A ,Jie S ,Nouman A , et al.Comparative CFD modeling of a bubbling bed using a Eulerian–Eulerian two-fluid model (TFM) and a Eulerian-Lagrangian dense discrete phase model (DDPM)[J]. Powder Technology,2021,383418-442.

[15] Y. Tsuji, T. Kawaguchi, T. Tanaka, Discrete particle simulation of two-dimensional fluidized bed, Powder Technol.77(1993)79-87.

[16] Vlasak ,Pavel,Kysela , et al.FLOW STRUCTURE OF COARSE-GRAINED SLURRY IN A HORIZONTAL PIPE[J].Journal of Hydrology and Hydromechanics,2012,60(2):115-124.