

PAPER • OPEN ACCESS

Numerical simulation of coarse particle pipeline transportation based on Eulerian Lagrangian model

To cite this article: Sibow Wang *et al* 2020 *J. Phys.: Conf. Ser.* **1600** 012005

View the [article online](#) for updates and enhancements.

You may also like

- [Localized Corrosion of 7xx.x Al Alloy Castings Grain-Refined with Al-5Ti-1B Master Alloy](#)
J. Kramp, S. Shankar and J. R. Kish
- [Effects and optimization of ratio of particle size grading on compaction density of calcined coke particles](#)
Wenyuan Hou, Hesong Li, Mao Li et al.
- [A four-way coupled Euler—Lagrange approach using a variational multiscale method for simulating cavitation](#)
Georg Hammerl and Wolfgang A Wall



ECS
The
Electrochemical
Society
Advancing solid state &
electrochemical science & technology

DISCOVER
how sustainability
intersects with
electrochemistry & solid
state science research

Numerical simulation of coarse particle pipeline transportation based on Eulerian Lagrangian model

Sibo Wang^{1, a}, Xuewen Ding^{1, b} and Jin Wang^{2, *}

¹School of Energy and Power Engineering, Wuhan University of Technology, Wuhan 430063, China

²Rizhao Polytechnic, Rizhao, China

*Corresponding author e-mail: 568857508@qq.com, ^a565380793@qq.com,

^b1370533273@qq.com

Abstract. In this paper, based on the Eulerian Lagrangian model, the non-phase change flow process of the coarse particle slurry in the horizontal pipeline is studied. The particle motion state is tracked, the dynamic characteristics of the coarse particle slurry in the pipeline transportation are explored, and the resistance characteristics of the pipeline and the operation stability under different conditions are analyzed. The comparison of the results of velocity distribution, concentration distribution and pressure curve with the empirical formula and experimental data shows that the simulation results are in good agreement with the calculation results of wasp formula and the experimental values of Roco&Balakrishnam, and the simulation can better describe the flow characteristics in the basin. In addition, the flow characteristics of particles are further analyzed according to the key factors such as the flow stability, the transition process and the critical deposition velocity of particles in different positions of the pipeline.

1. Introduction

The solid-liquid two-phase flow is often called slurry. The slurry pipeline transportation with powder or sand particles is very common in the long-distance transportation of bulk materials [1]. Among them, the particle size of powder particles is between 0.002-0.06 mm, which belongs to the category of fine particles in transportation, while the particle size of sand particles is between 0.02-2 mm, and larger than 0.2 mm, which is considered as coarse particle transportation [2]. In the process of transportation, the slurry has high turbulence effect in the high velocity basin, and the solid particles are almost evenly distributed in the whole pipeline section. However, the decrease of flow velocity will lead to the increase of particle concentration at the bottom of the pipeline and the complexity of transportation. When the flow velocity decreases to a certain value, the coarse particles will settle to form a dense sliding riverbed, which will eventually become a fixed riverbed after continuous accumulation. At this time, the mud velocity is called the critical deposition velocity [3]. The formation of bed will cause the pipeline to wear and block. In addition, the complex movement state of coarse particles in the pipeline at low speed will also aggravate the instability of transportation. Therefore, a complete description of its pipeline transportation flow state is very important for the study of stable transportation of coarse-grained mud.



At present, the research on pipeline water transportation focuses on the use of Eulerian model to describe the state of solid-phase flow and Reynolds average Navier Stokes method to simulate the turbulent properties of carrier fluid [4]. Ling et al [5] proposed a simplified three-dimensional algebraic sliding mixing (ASM) model combined with K - ε turbulence model for the numerical calculation of mud flow. This model can only predict the average pressure gradient above the critical deposition velocity. Ekambara et al. [6] Based on the theory of particle flow mechanics, used ANSYS-CFX to simulate the slurry transportation process in horizontal pipeline, and obtained the simulation results consistent with the experimental data, but it is only limited to the transportation of fine-grained slurry. Kaushal et al. [7] used the Eulerian Eulerian model to simulate the slurry flow of high concentration dispersed particles, and predicted the pressure drop and concentration distribution of continuous phase fairly accurately, but failed to capture the flow of single particles. In a word, the method based on the Eulerian model can accurately capture the flow situation in the whole basin, but it cannot accurately describe the interaction between particles and liquid, nor can it accurately display the flow characteristics of discrete particles.

According to the characteristics of coarse particles, Lagrangian model is used to track the independent particles, and the simulation process can reproduce the behaviour of particles in the pipeline to the greatest extent. After coupling the solver with the liquid phase under the Eulerian model, the particle dynamics characteristics are further refined by using the high-order data, and a complete pipeline transportation simulation scheme of coarse particle slurry under the Eulerian Lagrangian framework is proposed.

2. Mathematical model

In the Eulerian Lagrangian model, the single particle trajectory is solved by the Lagrangian method, while the fully developed liquid turbulence is solved on the background of the Eulerian grid. The two phases are fully coupled by the condition of uniform volume fraction and exchange momentum.

2.1. Conservation equation

In the case of complete turbulence, the K - ε equation of the turbulence model should be considered first, while in the case of turbulence, the drag force and the turbulent diffusion force have a significant impact on the numerical calculation results of each phase, so the drag force and the turbulent diffusion force equation should also be considered [8].

In addition, the flow process satisfies the conservation of momentum and mass, but the Lagrangian method is introduced to study particles. In order to solve the equation of motion of the fluid in the liquid instead of solving the flow around a single particle, the volume filter operator is applied to the Navier Stokes equation, so that the point variables (fluid velocity and pressure, etc.) can be replaced by smoother, locally filtered fields. The conservation equations of mass and momentum is modified.

(1) Turbulence model

The k - ε equation is suitable for most engineering turbulence models, and its expression is as follows:

The transport equation of turbulent kinetic energy k,

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{u_t}{s_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \quad (1)$$

The transport equation of dissipation rate ε of turbulent energy consumption,

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{u_t}{s_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (2)$$

Where $C_{1e}=1.44$; $C_{2e}=1.92$; $s_k=1.0$; $s_e=1.3$

(2) Equations of drag force and turbulent diffusion force

The momentum transfer between phases is mainly dominated by drag force, and its equation is as follows:

$$F_D = \frac{3}{4} C_D \alpha_s \rho_l \frac{1}{d_p} |u_l - u_s| (u_l - u_s) \quad (3)$$

The model of turbulent diffusion force in fluent software is based on the average value of Favre of inter phase traction force. The equation is as follows:

$$F_{TD} = \frac{3}{4} \frac{C_D}{d_p} \frac{\nu_{tc}}{\sigma_{tc}} \alpha_l \rho_s (u_l - u_s) \left(\frac{\nabla \alpha_l}{\alpha_l} - \frac{\nabla \alpha_s}{\alpha_s} \right) \quad (4)$$

Where $\sigma_{tc}=0.9$.

(3) Volume filtration continuity equation

The volume filtration continuity equation is as follows:

$$\frac{\partial}{\partial t} (\varepsilon_f \rho_f) + \nabla \cdot (\varepsilon_f \rho_f u_f) = 0 \quad (5)$$

(4) Momentum equation

The momentum equation is as follows:

$$\frac{\partial}{\partial t} (\varepsilon_f \rho_f u_f) + \nabla \cdot (\varepsilon_f \rho_f u_f \cdot u_f) = \nabla \cdot (\tau - \mathbf{R}_u) + \varepsilon_f \rho_f g - F^{inter} + F^{mfr} \quad (6)$$

2.2. Particle flow model

Lagrangian model is used to track particles, mainly to solve the displacement and stress of single particles. The displacement of a single particle indicated by subscript p is calculated by Newton's second law of motion, which is

$$m_p \frac{du_p}{dt} = f_p^{inter} + F_p^{col} + m_p g \quad (7)$$

Where $m_p = \pi \rho_p d_p^3 / 6$.

In addition, the experimental results show that during the solid-liquid coupling, there is no near wall lift for the finer particles, while the near wall lift related to the coarser particles is related to Magnus effect, Saffman force or other lifting forces. Although the lift effect may have some effect on the average motion of particles, these effects are not enough to affect the coupling process. In addition, the other influencing factors of interphase exchange, including the added mass term and basset effect term, are ignored, and only the non-negligible effects caused by the pressure gradient force and viscous stress of volume filtration fluid are considered [8]. The angular momentum of particles is only due to particle collisions as follow:

$$I_p \frac{d\omega_p}{dt} = \sum_j \frac{d_p}{2} n \times f_{t,j \rightarrow p}^{col} \quad (8)$$

Where Tangential component of collision force of particle j acting on particle p , and $I_p = \frac{m_p d_p^2}{10}$.

In order to correctly calculate the collision situation without too small time step, when Fluent is used for simulation, $\tau_{col} = 15\Delta t$. In addition, the collision near the wall is treated as a particle with infinite mass and zero radius, which is

$$f_{t, b \rightarrow a}^{col} = -\mu_f \left| f_{n, b \rightarrow a}^{col} \right| t_{ab} \quad (9)$$

Where $t_{ab} = \frac{u_{ab, t}}{|u_{ab, t}|}$, and $u_{ab, t} = u_{ab} - u_{ab, n}$.

3. Numerical scheme

3.1. Physical model

ICEM CFD was used to establish spiral welded steel pipe with diameter $d = 100\text{mm}$ and length $L = 3\text{M}$ ($> 20d$) in order to meet the full development of the basin and to meet the experimental conditions. In the Eulerian Lagrangian model, because particles enter the computational domain in discrete state, in order to fully capture the spatial scale range associated with turbulence, a 25 layer boundary layer with exponential growth rate of 1.2 is established, and the grid size is about the same as the average particle size. In addition, the height of the first floor from the wall is expressed as the dimensionless parameter Y^+ , where $y^+ < 5$, in order to obtain better calculation results. The total number of generated grids is 586731. The section view of the pipe mesh is shown in Fig. 1

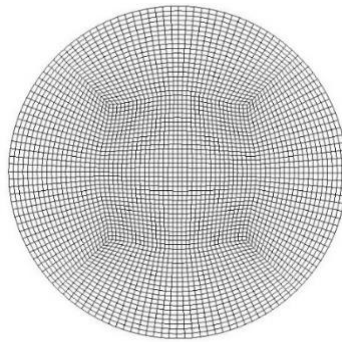


Figure 1. Cross section of pipe mesh

3.2. Boundary condition

The periodic boundary conditions expand along the X direction. The velocity distribution at the entrance is defined to simulate the entrance more accurately, so that the boundary layer has developed completely at the entrance. The power law of seven component energy is used to describe the entrance profile [9], and its distribution is as follows:

$$U = W_{max} \left(1 - \frac{r}{R_{max}} \right)^{\frac{1}{7}} \quad (10)$$

In this simulation, two groups of working conditions are carried out to observe the simulation, $W_{max} = 5m/s$ in case A, $W_{max} = 1.2m/s$ in case B, $W_{max} = 1.2m/s$, R_{max} is the radius of the pipe, r is the distance from the center line of the pipe, defined as $r = \sqrt{x^2 + y^2}$, and the initial velocity of the foundation phase and the discrete phase are the same

3.3. Numerical solution and convergence scheme

In order to study the detailed physical information of dredged materials in horizontal pipelines, the NGA framework for the calculation of high-order turbulence is used to realize the finite difference scheme of mass, momentum and kinetic energy [10]. Navier Stokes equation is used to solve the second-order spatial precision mesh with convection and viscosity terms. In addition, the second-order accurate semi implicit crank Nicolson method is used to advance the time. The time step is constant 0.001s, and the time distribution of flow variable is calculated in 100s.

The distribution of particles is based on the liquid phase. For each particle, its position, velocity and angular velocity are solved by the second-order Runge Kutta scheme. When the particle response time becomes smaller than the simulation time step, the sub step is further used to ensure the stability, and the coupling between the liquid and solid particles occurs in the form of volume fraction. In order to interpolate the fluid variable to the particle position, the second-order trilinear interpolation scheme is used. The particle data are effectively pushed back to the "O" grid by using the two-step simultaneous softening diffusion operation. This strategy has been proved to be convergent under mesh refinement. For each case, the simulation runs long enough to reach a statistical standstill. The results are collected after about $\tau = 50$, where dimensionless time $\tau = tV_f/D$.

4. Results

Fig. 2 and Fig. 3 are the profiles of particle concentration and liquid velocity of the pipeline cross section at the axial position separated along the 0.3m interval under condition a respectively. It can be observed that there are obvious differences in the particle concentration distribution and liquid velocity distribution between the first and sixth axial positions. The contour map from the seventh axial position to the eleventh axial position is almost the same, so the flow pattern in this area has been fully developed. In order to ensure the accuracy and reliability of the analysis results, the concentration and velocity simulation values mentioned later are obtained at the tenth section of the pipeline.

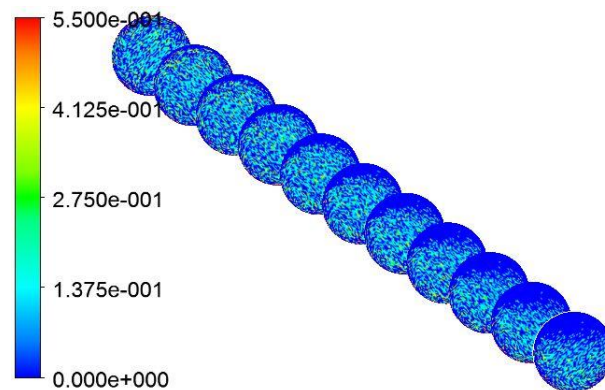


Figure 2. Particle concentration distribution of the cross section

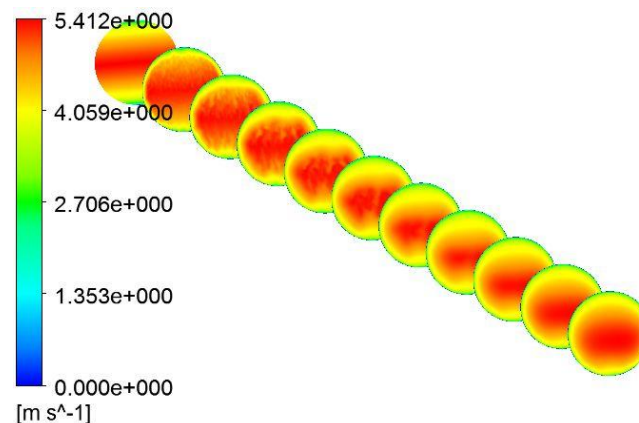


Figure 3. Liquid velocity distribution of the cross section

4.1. Distribution of overflow logistics to diffusion

The particle concentration distribution diagram of condition a and condition B is shown in Fig. 4, Fig. 5, Fig. 6 and Fig. 7. The concentration distribution shows asymmetry. Due to the particle deposition, a large particle concentration is observed in the lower part of the vertical axis of the center, while a small particle concentration is observed in the upper part of the pipeline. In both simulations, the flow consists of different regions controlled by hydrodynamics.

To facilitate analysis and discussion, these different layers are described by different dashed lines in the figure. The flow region in case B can be divided into three parts: region 1, region 2 and region 3. The particle concentration distribution shows that the particles are almost completely located in the lower half of the tube. Area 1 is the range of $y' = 0$ to $y' = 0.095$, which corresponds to the fixed bed at the bottom of the pipe. The particles in the bed are piled up intensively and experience continuous contact, and the particle concentration is almost constant, so it is not conducive to the average movement of the fluid. Area II is directly above the bed and extends to the watershed with particle concentration of about 25%, corresponding to the area with $y' = 0.095$ to $y' = 0.172$. The region is composed of strongly colliding particles and highly turbulent liquid. Particles can be lifted from the bed by strong turbulent vortex, or ejected by other particles colliding with the bed. Finally, zone 3 is located away from the bed, with a fairly small average particle concentration and particles remaining suspended due to fluid velocity fluctuations.

In case a, the difference between flow regions is not as obvious as that in case B. In this case, there is no fixed bed at the bottom of the pipeline, so the flow field is only divided into two regions: region two and region three. For the convenience of observation and comparison, the transition interface between zone 2 and zone 3 under condition a corresponds to zone 1 and zone 2 under condition B, which is $y' = 0.095$.

In comparison with the experimental data, condition a is in good agreement with Roco & balakrishnam. In addition, the trend of particle concentration observed in the second area of the bottom of the pipeline is faster than that of the simulation results, and the simulation results are reversed near the wall. The difference is due to the error caused by the inaccuracy of the near wall modeling, and the result may be caused by the neglect of the lifting force.

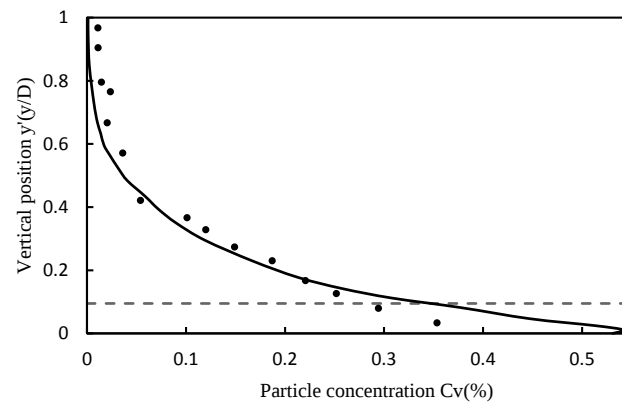


Figure 4. Particle concentration distribution curve in case A

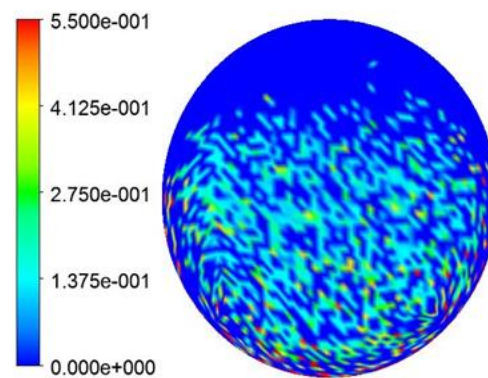


Figure 5. Cloud of particle concentration distribution in case A

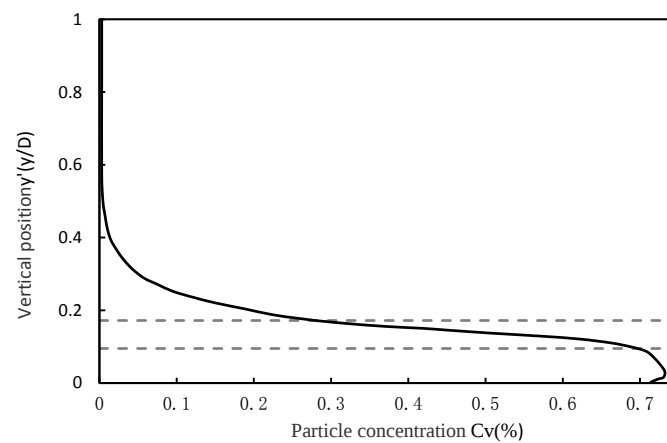


Figure 6. Particle concentration distribution curve in case B

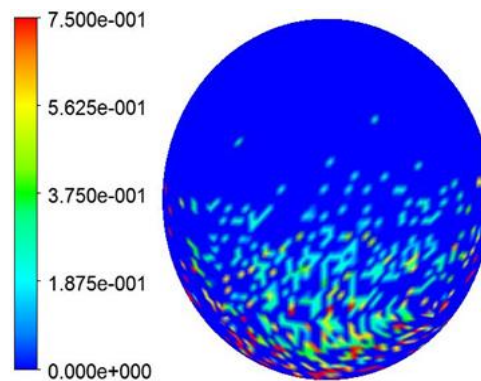


Figure 7. Cloud of particle concentration distribution in case B

4.2. Radial diffusion distribution of overflow

The speed curves of condition a and condition B are shown in Fig. 8 and Fig. 9. Due to the suspension of fine particles and the sedimentation of coarse particles, the particle velocity is asymmetric in the radial vertical direction, and the upper velocity is slightly larger than the lower velocity, and the velocity distribution in condition a is in good agreement with the experimental results of Roco & balakrishnam.

In case B, the particle velocity in area I is very small and almost constant, and it begins to increase in area II and the trend is consistent with case a. In zone 1, at low flow velocity, turbulence is no longer dominant at the bottom of the pipe, and the shear stress of the flowing liquid cannot overcome the submerged weight of the particles, so the particles settle and form a fixed bed.

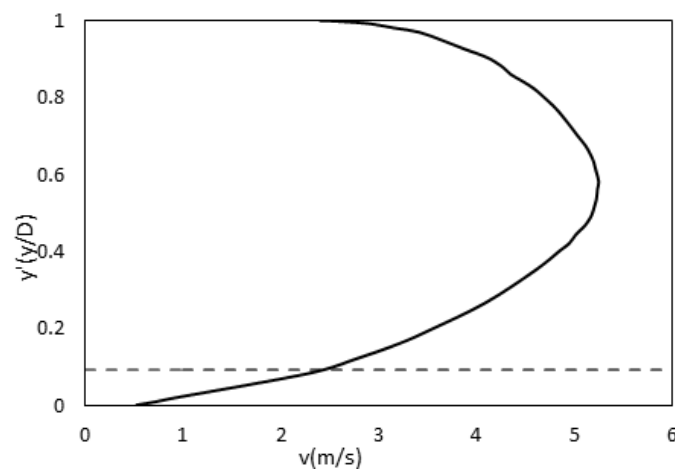


Figure 8. Particle velocity distribution curve in case A

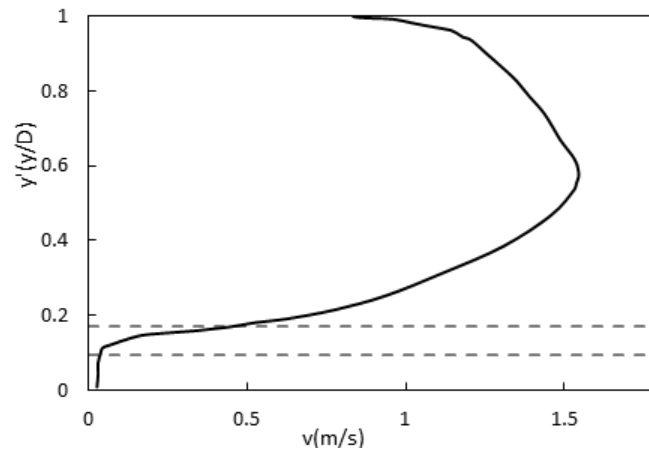


Figure 9. Particle velocity distribution curve in case B

4.3. High order statistical results

In order to further understand the particle behavior and related dynamic information, in order to better explore the flow situation of the pipeline, the full development area of the two conditions simulation is intercepted, and the higher-order statistical data is extracted for analysis.

(1) Variance analysis of particle volume fraction

The variance of particle volume fraction is shown in Fig. 10. For case B, the maximum concentration fluctuation is located at the interface between zone 2 and zone 3. In region 3, the fluctuation decays rapidly due to the suspension of particles and the dilution of flow. Area 1 is a fixed bed area. The Fig. 11 shows that the variance value of its volume fraction is very small, which indicates that there is no obvious concentration fluctuation in this area. It is further confirmed that particles accumulate intensively in this area and maintain continuous rigid contact. In contrast, the fluctuation of concentration in case A cannot be ignored at the bottom of the pipe, which indicates that the flow at the bottom of the pipe is dominated by turbulence at a higher speed

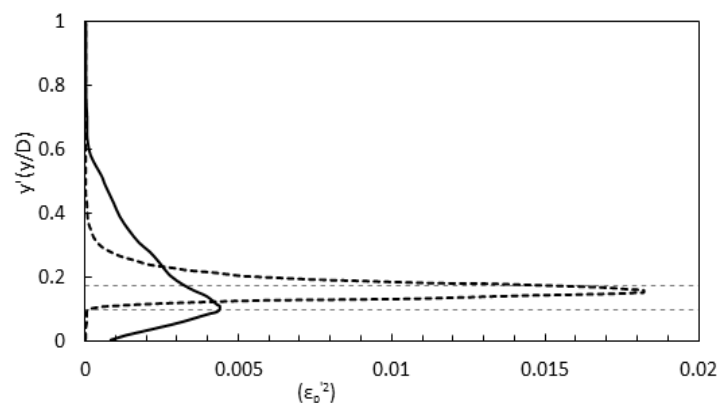


Figure 10. Particle concentration variance profile curve

Generally speaking, the concentration fluctuation range of case A and B reaches the peak value when the transition area develops to the suspension area, which may be caused by the strong collision of particles at the interface of region 2 and region 3, and the flow has the property of high collision shear flow, which is also an important factor affecting the stability of pipeline transportation. In addition, it is found that the fluctuation of particles is more obvious when operating below the critical deposition speed, which indicates that the lower operation speed makes the working condition more unstable. Therefore,

the pipeline transportation of coarse particles under the condition of lower than the critical deposition speed should be avoided as far as possible in practice.

(2) Particle velocity variance analysis

The variation of velocity variance in different directions of particles under condition A and condition B is shown in Fig. 11 and Fig. 12. The thick solid line represents the particle velocity variance along the axial direction, the thin solid line and the dotted line represent the particle velocity variance along the radial vertical direction and the radial horizontal direction respectively. It can be seen from the above that the velocity fluctuation of the two working conditions reaches the peak value at the interface from the transition area to the suspension area and close to the top of the pipe. The peak value at the junction of the transition area is caused by the high turbulence intensity in the transition area and the high impact shear flow of particles, while the peak value at the top of the pipe is caused by the shear action of the wall. In addition, after entering the suspension area, the particle concentration decreases, and the suspension and flow dilution properties of a few particles make the velocity fluctuation continuously decrease, which also shows that the flow near the center of the pipe is the most stable.

In terms of specific numerical value, the fluctuation range of velocity along the axial direction of the two conditions is the largest, which is related to the particles shooting into the pipeline along the axial direction. The velocity fluctuation range along the radial vertical direction is larger than that along the radial horizontal direction, which shows that the velocity distribution under the two conditions has certain symmetry in the radial horizontal direction. In addition, Fig. 12 shows that there is almost no velocity fluctuation at the bottom of the pipe under case B, and the side also reflects the flow state of particles with very low Reynolds number after dense accumulation at the bottom of the pipe, which further proves the formation of fixed bed at the bottom of the pipe.

The overall average value of velocity variance in condition a is far lower than that in case B, which indicates that the stability of operation in condition a is higher than that in case B, and the particle flow in case B is affected by more factors, which aggravates the instability in the transportation process.

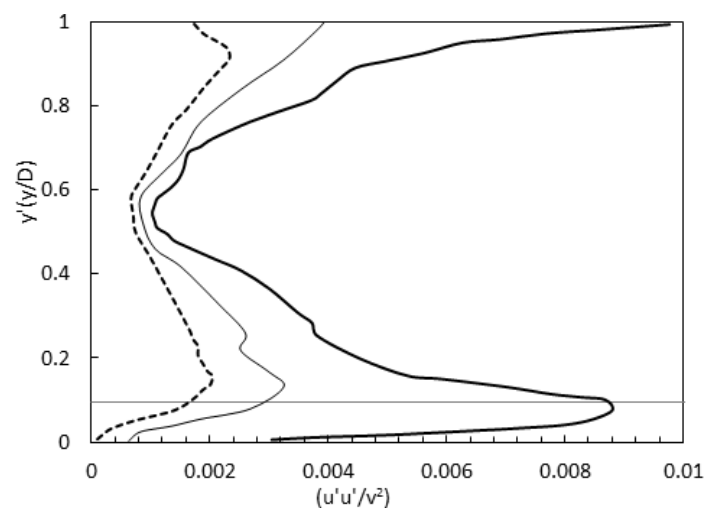


Figure 11. Particle velocity variance profile curve in case A

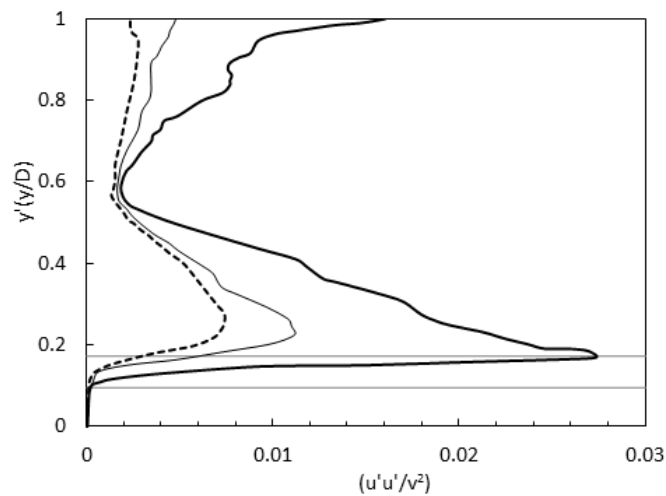


Figure 12. Particle velocity variance profile curve in case B

4.4. Pressure drop

The experimental pressure drop curves of condition a, condition B and Roco & balakrishnam are shown in Fig. 13. In order to ensure that the curve is the curve of the fully developed area of the basin, the average pressure of five sections after the seventh section mentioned above is selected for simulation. It can be observed that the average pressure drop in condition a is 610 PA / m, while the average pressure drop in the experiment is 666.3 PA / m, which may be caused by the error caused by neglecting the viscosity flow on the pipe surface in the simulation calculation or the inaccurate modeling near the wall. In addition, the average pressure drop of case B is 813 PA / m. It can be seen that there is great resistance in the basin when operating near or below the critical deposition speed, so this kind of condition is very unfavorable to the transportation of particle pipeline in the actual project.

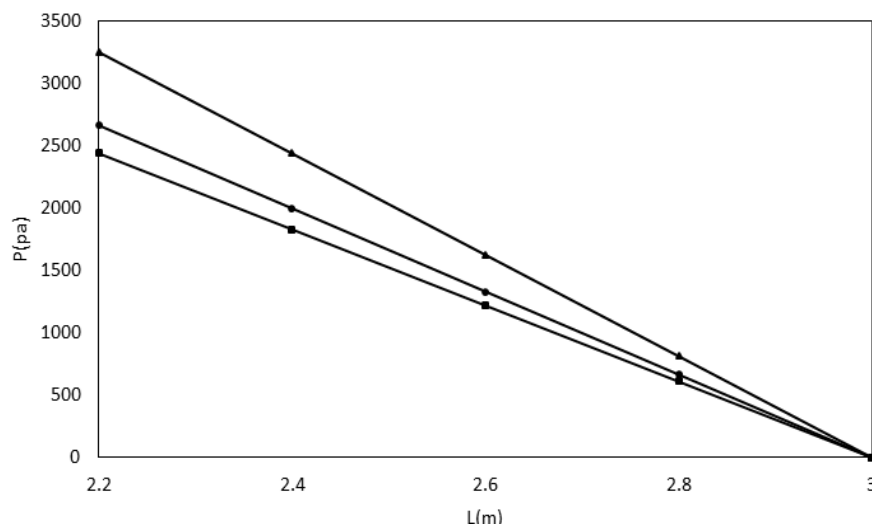


Figure 13. Mean pressure curve along horizontal centreline

5. Conclusion

In this paper, the simulation results are consistent with the experimental results under the same initial conditions as the experiment, which verifies the correctness of the Eulerian Lagrangian model. Through the simulation results under different working conditions, it is further shown that the simulation of Euler

Ian Lagrangian model can describe the flow of solid-liquid two-phase flow with coarse particles in the pipeline transportation process more completely. On this basis, the following conclusions are drawn:

(1) By changing the velocity variable, observe and compare the simulation results. When the velocity is lower than the critical deposition velocity, the particles accumulate at the bottom of the pipeline, the concentration of particles at the bottom of the pipeline increases sharply, and the velocity drops sharply, forming a fixed bed area.

(2) Through the statistical analysis of the high-order data, it is found that the fluctuation range of particle velocity and particle volume fraction reaches the peak value at the top near the wall and at the junction of the transition area and the suspension area, and the minimum value is near the center of the pipeline, so as to obtain the relatively stable area in the pipeline transportation. In addition, when the velocity is lower than the critical deposition velocity, the average fluctuation range of particle concentration and velocity is large, and the transportation stability is very poor under this condition, which is easy to cause pipeline transportation failure.

(3) By comparing the experimental value, simulation value and empirical formula of unit pressure drop, the simulation result can better predict the pressure drop change compared with wasp model. Combined with the above concentration and velocity distribution, a fixed bed is formed at the bottom of the pipeline under the condition of lower than the critical deposition velocity, which reduces the effective pipe diameter in the transportation process, increases the resistance along the pipeline, and also increases the risk of pipeline blockage. Therefore, it should be avoided as much as possible in the actual project.

Acknowledgments

This work supported by self-determined and innovative research funds of WUT.

References

- [1] Televantos Y, Shook C, Streat M, et al. 1979, Flow of slurries of coarse particles at high solids concentrations [J]. *Canadian Journal of Chemical Engineering*, 57 (3): 255 - 262.
- [2] Deen NG, Annaland MVS, Hoef MAVD, et al. 2007, Review of discrete particle modeling of fluidized beds [J]. *Chemical Engineering Science*, 62 (1–2): 28 - 44.
- [3] Azimi AN H, Zhu D Z, Rajaratnam N. 2012, Computational investigation of vertical slurry jets in water [J]. *International Journal of Multiphase Flow*, 47 (3): 94 - 114.
- [4] Ling J, Skudarnov P V, Lin C X, et al. 2003, Numerical investigations of liquid–solid slurry flows in a fully developed turbulent flow region [J]. *International Journal of Heat & Fluid Flow*, 24 (3): 389 - 398.
- [5] Ekambara K, Sanders R S, Nandakumar K, et al. 2009, Hydrodynamic Simulation of Horizontal Slurry Pipeline Flow Using ANSYS-CFX [J]. *Industrial & Engineering Chemistry Research*, 48 (17): 8159 - 8171.
- [6] Kaushal D R, Thinglas T, Tomita Y, et al. 2012, CFD modeling for pipeline flow of fine particles at high concentration[J]. *International Journal of Multiphase Flow*, 43:85 - 100.
- [7] Balachandar S, Eaton J K. 2010, Turbulent Dispersed Multiphase Flow [J]. *Advances in Mechanics*, 42 (1): 111 - 133.
- [8] Kaushal D R, Tomita Y. 2007, Experimental investigation for near-wall lift of coarser particles in slurry pipeline using γ -ray densitometer [J]. *Powder Technology*, 172 (3): 177 - 187.
- [9] Cundall P A, Strack O D L. 2008, a discrete numerical model for granular assemblies [J]. *Géotechnique*, 30 (30): 331 - 336.
- [10] Li Y L, Tung K L. 2008, CFD simulation of fluid flow through spacer-filled membrane module: selecting suitable cell types for periodic boundary conditions [J]. *Desalination*, 233 (1): 351 - 358.