Study on the length of pipe inlet section of pulp fluid based on CFD

Song Gao1, Xinwu Du1, Shengjie Jin1, Qiaohong Liu2 and Hao Ma1

Abstract
In the paper industry, to determine the installation positions of concentration sensor and flow sensor of pulp pipe transportation system, the length of pipe inlet section of pulp fluid must be determined. In order to solve the problem that it is difficult to determine the length of pipe inlet section of pulp fluid, the paper presents a method to determine the length of pipe inlet section of pulp fluid, and the relationship between the length of inlet section and pulp fluid parameters is studied by this method. Firstly, CFD (Computational Fluid Dynamics) method is applied to obtain flow velocity and solid phase concentration distribution data of pulp fluid at different axial positions in the pipe. Then, Pearson correlation coefficient method is applied to analyze these data. Finally, the length of pipe inlet section of pulp fluid is determined according to the obtained correlation coefficient, and the relationship between the length of pipe inlet section of pulp fluid and initial average flow velocity and solid phase concentration is obtained. The conclusion shows that the method can well obtain the length of pipe inlet section of pulp fluid, which provide some theoretical basis for the design of pulp pipe transportation system.

Keywords
Pulp fluid, length of pipe inlet section, CFD, correlation coefficient

Date received: 7 May 2020; accepted: 24 July 2020

Introduction
In the paper industry, pulp fluid concentration and pulp flow velocity are the two most frequently measured physical quantities. The measurement accuracy of pulp concentration and pulp flow velocity is very important for automatic control of production process, ensuring the quality of pulp and paper, reducing the consumption of raw materials, improving production efficiency and saving production cost.1–3 In recent years, with the application of high-speed paper machines, papermaking production lines have also put forward corresponding requirements for the rapidity and accuracy of pulp concentration and pulp flow velocity measurement.4 So, when designing the pulp pipeline transportation system, it is necessary to ensure that the concentration and flow velocity of pulp fluid can obtain sufficiently accurate measurement results. Generally, the concentration and flow velocity of pulp fluid are measured by installing sensors on the pulp transportation pipeline.5 In the case of straight pipe, the installation position of the sensor on the pipe is determined according to the length of pipe inlet section of pulp fluid.

Under the conditions of without considering the bent pipe or the blocking body inside the pipeline and the secondary flow of the fluid, the flow of pulp fluid in the straight pipe is divided into two areas, which are the pipe inlet section and the pipe fully developed section. The area from the inlet of pipe to the stable flow state of the fluid is pipe inlet section, and the pipe area downstream of inlet section is pipe fully developed section. In the pipe inlet section, the solid phase concentration distribution and flow velocity distribution of pulp fluid are in a state of continuous change, after entering the pipe fully developed section, the solid phase concentration distribution and flow velocity distribution of pulp fluid reach the stable state. In order to ensure the measurement accuracy, the pulp concentration and flow sensors need to be installed on the fully developed section of pulp pipe.
the pipe, so the length of pipe inlet section of pulp fluid must be determined.

At present, the main method for determining the installation position of the sensor in the papermaking industry is to set a long straight pipe upstream of the installation position of the sensor to ensure that the installation position of the sensor is at the pipe fully developed section. This method is only a simple method to determine the installation position of the sensor, it lacks theoretical support and cannot give a definite value of the length of the straight pipe. If the straight pipe is not long enough, it may be less than the length of pipe inlet section of pulp fluid, which inevitably causes measurement errors. If the length of the straight pipe is too long, it will occupy more production sites and waste resources. Therefore, theoretically solving the problem of how to determine the length of pipe inlet section of pulp fluid is of certain significance to the design of the pulp pipeline transportation system in the paper industry.

At present, there are many research results on the pipe inlet section of fluid. Mohanty A K et al. studied the length of pipe inlet section of low Reynolds number fluid through theoretical calculation and a large number of experiments. Zanoun et al. studied the length of inlet section of single-phase flow by experimental method, and proposed an approximate method to determine the length of pipe inlet section of single-phase fluid. Len et al. obtained the law of the length of pipe inlet section and Reynolds number of high Reynolds number fluid through experiments, and obtained the range of the length of pipe inlet section. These studies mainly use theoretical calculation and experimental methods to determine the length of pipe inlet section for single-phase fluid, and some empirical formulas are summarized to determine the relationship between the length of pipe inlet section and the Reynolds number of fluid. These methods and research results are of great significance in determining the length of pipe inlet section of single-phase fluid in the engineering field. However, the pulp fluid is slurry fluid and belongs to liquid-solid two-phase flow. For the problem of how to determine the length of pipe inlet section for slurry fluid, it is difficult to obtain accurate conclusions by using experimental methods or empirical formulas due to the limitations of model size, flow field disturbance and measurement accuracy.

The equation of motion of fluid can be described by N-S equation (Navier-Stokes equation). By solving the N-S equation, the exact solution of the length of the inlet section of slurry fluid pipeline can be obtained. However, the N-S equation is a nonlinear partial differential equation, which is very difficult to solve. In recent years, with the wide application of some new methods and numerical analysis methods for solving non-linear problems in engineering, many research results have been obtained by solving fluid motion equations or numerical analysis methods to study fluid flow problems. Compared with the experimental method, the method of solving the equation of motion or numerical analysis to study the fluid flow problem has stronger applicability and higher accuracy.

As a method to solve the numerical solution of partial differential equations describing fluid motion, Computational Fluid Dynamics (CFD) method has been widely used in the research of flow. With the development of CFD, great progress has been made in the research of slurry fluid. The CFD method is a discrete approximate calculation method, and it can be used to obtain the numerical solution of fluid flow field problems. Although the CFD method cannot obtain the analytical solution of the fluid flow problem, the error between the numerical solution obtained by this method and the real value is very small, which can well meet the needs of engineering. Hsu et al. established the multiphase flow model of slurry fluid, which can describe the solid phase concentration distribution of slurry fluid. Lin et al. established a high concentration slurry fluid model, which can calculate the concentration and flow velocity distribution of high concentration slurry fluid. Ling et al. proposed a simplified multiphase flow model to calculate the flow of slurry fluid and obtain numerical solutions, which can predict the flow pressure drop, concentration and flow velocity distribution of slurry fluid. Shinchichi and He et al. believe that Euler-Euler multiphase flow model can be used to simulate and calculate the general problems of multiphase flow, and they used Euler-Euler multiphase flow model method to simulate the flow of slurry fluid in the straight pipe. In these studies, the CFD multiphase flow method is used to establish the dynamic model of slurry fluid, and the computer is used to simulate the flow of slurry fluid, so as to obtain the numerical solution of slurry fluid flow problem at a limited number of discrete points, which can well solve the pipe flow problem of slurry fluid. These studies show that the CFD method can overcome the shortcomings of theoretical analysis and experimental methods, and it is appropriate to solve the flow problem of slurry fluid. The pulp fluid is a kind of slurry fluid, so the CFD method is suitable for the study of the length of pipe inlet section of pulp fluid in this paper.

The innovation of this work are mainly reflected in the CFD method is proposed to determine the length of pipe inlet section of pulp fluid, and illustrates the specific process of applying this method through simulation experiments and data analysis, which provides a convenient and low-cost method for determining the installation positions of pulp concentration sensor and flow sensor on the pipeline. This method is simple in principle and easy to solve.

The main work and contributions of this paper are as follows:

1. The CFD method is proposed to determine the length of pipe inlet section of pulp fluid, the correctness of the method and results are verified by simulation experiments and data
analysis, and the specific process of applying the method is explained.

(2) The low that the length of pipe inlet section of pulp fluid is changed by the change of average flow velocity and solid phase concentration of pulp fluid is summarized.

(3) The method and conclusion of this paper provide theoretical support for how to determine the installation position of pulp concentration sensor and flow sensor on the pipeline, which is of certain significance to the design of pulp pipeline transportation system in paper industry.

Mathematical model

In this paper, the Euler-Euler multiphase flow model is used to establish the two-phase flow dynamic model of pulp fluid, and the RNG turbulence equation\(^21\) is used to describe the turbulent flow of pulp fluid.

To establish a mathematical model of pulp flow, the following conditions are required:

(1) The solid phase is a continuous fluid, the liquid phase is an incompressible fluid, and the physical properties of each phase are constant.

(2) There is no phase change in both solid and liquid phases, and cavitation phenomenon in the field is not considered.

The liquid phase of pulp fluid is water and the solid phase is pulp fiber, so the above conditions can be met.

Equation of motion of pulp fluid

The equation of motion of pulp fluid is established in Euler coordinate system as follows:

\[
\frac{\partial \phi_L U_L}{\partial t} + \frac{\partial}{\partial x_i} (\phi_L U_L U_L) = 0 \quad (1)
\]

\[
\frac{\partial \phi_S}{\partial t} + \frac{\partial}{\partial x_i} (\phi_S U_S) = 0 \quad (2)
\]

Equations (1) and (2) can be authenticate by reference.\(^22\) Where \( \phi_L \) is the volume fraction of liquid phase, \( \phi_S \) is the volume fraction of solid phase, \( U_L \) is the velocity component of liquid phase, \( U_S \) is the velocity component of solid phase, and \( x_i \) is the coordinate component. If the liquid phase density and the solid phase density are \( \rho_L \) and \( \rho_S \) respectively, equations (3) and (4) can be obtained according to equations (1) and (2):

\[
\nabla (\phi_L \rho_L U_L) = 0 \quad (3)
\]

\[
\nabla (\phi_S \rho_S U_S) = 0 \quad (4)
\]

Equation (3) is a liquid phase continuous equation and equation (4) is a solid phase continuous equation, the momentum equations is:

\[
\nabla (\phi_L \rho_L U_L U_L) = -\phi_L \nabla p + \nabla \delta_L + B_{LS} (U_S - U_L) + \phi_L \rho_L g \quad (5)
\]

\[
\nabla (\phi_S \rho_S U_S U_S) = -\phi_S \nabla p - \nabla p_{SS} + \nabla \delta_S + B_{LS} (U_L - U_S) + \phi_S \rho_S g \quad (6)
\]

where \( \nabla p \) is the shared pressure of two fluid phases, \( \nabla p_{SS} \) is the solid phase pressure generated by pulp fibers colliding with each other, \( \delta_L \) and \( \delta_S \) are the stress tensors of liquid phase and solid phase respectively, \( g \) is the gravitational acceleration, and \( B_{LS} \) is the momentum transfer coefficient between liquid and solid phases. The value of \( B_{LS} \) is related to the volume fraction of liquid phase \( \phi_L \), and the detailed calculation method of \( B_{LS} \) is given in literature.\(^23\) Only the liquid and solid phases of pulp fluid are considered, therefore:

\[
\phi_L + \phi_S = 1 \quad (7)
\]

then, \( \delta_L \) and \( \delta_S \) can be determined by equations (8) and (9):

\[
\delta_L = \phi_L \mu_L [\nabla U_L + (\nabla U_L)'] - \frac{2}{3} \phi_L \mu_L (\nabla U_L) I \quad (8)
\]

\[
\delta_S = \phi_S \mu_S [\nabla U_S + (\nabla U_S)'] + \phi_S \left( \lambda_S - \frac{2}{3} \mu_S \right) (\nabla U_S) I \quad (9)
\]

where \( \mu_L \) is the average effective viscosity of the liquid phase, \( \mu_S \) is the solid phase shear viscosity, \( \lambda_S \) is the solid phase volume viscosity, and \( I \) is the unit tensor.

Turbulent model of pulp fluid

The turbulent flow of pulp fluid can be described by RNG \( k - \varepsilon \) turbulence model.\(^21\) In general, the flow state of pulp fluid in the paper industry is turbulent, which needs to be simulated by using turbulence model. The standard \( K - \varepsilon \) model was proposed by Launder and Spalding\(^9\) and is currently the main tool for turbulence calculation in engineering. RNG \( k - \varepsilon \) model is an improvement on the standard \( K - \varepsilon \) model and has the following advantages:

a. RNG model adds an additional viscosity term to the N-S equation, and its calculation accuracy is higher.

b. The standard \( K - \varepsilon \) model is a high Reynolds number model, while RNG \( k - \varepsilon \) model can be used for fluids with lower Reynolds number.

These characteristics make RNG \( K - \varepsilon \) model more reliable and accurate than standard \( k - \varepsilon \) model. Therefore, we use RNG \( K - \varepsilon \) turbulence model.

Applying RNG method\(^24,25\) to N-S equation and introducing turbulence kinetic energy \( k \) and dissipation rate \( \varepsilon \), the following model can be obtained:

\[
\nabla (\rho_{LS} U_{LS} k) = \nabla [\alpha_0 (\mu_0 + \mu_e) \nabla k] + 2 \mu_S S^2 - \rho_{LS} \varepsilon \quad (10)
\]
Constants are:

\[ \nabla (\rho_{LS}ULS) = \nabla [\mu_0 + \mu_t] \nabla e] + 2c_1 \frac{e}{R} \mu_t S^2 \]
\[ - c_2 \rho_{LS} \frac{e^2}{R} - R \]

In equations (10) and (11), \( \rho_{LS} \) is the average density of pulp fluid, \( U_{LS} \) is the average speed of pulp fluid, \( \mu_0 \) is the kinematic viscosity of pulp fluid, \( \mu_t \) is the turbulent kinematic viscosity coefficient, \( S = \nabla U_{LS} \), \( c_1 \), \( c_2 \) are constants, according to literature, their values are: \( c_1 = 1.42 \), \( c_2 = 1.68 \), \( \alpha_t = \alpha_h = 1.39 \). \( R \) is the effect of average strain rate on dissipation rate \( \varepsilon \), according to literature, \( R = \frac{c_\mu e^2}{k} (1 - \eta / \eta_0) \)

\[ \eta = \sqrt{2Sk/\varepsilon}, \eta_0 \text{ is the typical value of } \eta \text{ in uniform shear flow, usually } \eta_0 = 4.38, \text{ the values of other constants are: } c_\mu = 0.085, \beta = 0.012. \]

Simulation experiment and data analysis

According to the above mathematical model, the CFD method can be applied to obtain the numerical solution of the flow parameters of pulp fluid in the pipeline. The method for determining the length of pipe inlet section of pulp fluid in this paper is as follows: a plurality of positions are sequentially selected in the axial direction of the pipe, and the flow velocity and concentration distribution of pulp fluid at these positions are obtained by CFD method, when the velocity and concentration distribution on all pipe areas downstream of a certain position reach a stable state, the upstream of this position is pipe inlet section, and the downstream of this position is pipe fully developed section.

Boundary conditions and pipe model

In this paper, Comsol Multiphysics\textsuperscript{27} is used for numerical simulation and without considering the influence of temperature and density changes on pulp fluid. The boundary conditions and initial conditions are set as follows:

1. Initial volume concentration of pulp fluid at the pipe inlet.

The range of pulp concentration in the papermaking industry is generally divided into low concentration pulp (\( \phi_s \approx 2\% \)), medium concentration pulp (2% \( \leq \phi_s \leq 6\% \)) and high concentration pulp (\( \phi_s \geq 6\% \)). Therefore, the initial volume concentration of pulp fluid under five conditions of 2%, 5%, 10%, 15%, and 20% is selected, and these concentration parameters can basically include the range of pulp concentration in the papermaking industry.

2. Initial average flow velocity of pulp fluid at the pipe inlet.

At present, the average flow velocity of pulp fluid \( U_{LS} \) is in the range of 0.05\( m/s \) ~ 10\( m/s \) in the pulp pipeline transportation system of the paper industry, so the initial average flow velocity of pulp fluid is set in nine cases of 0.05\( m/s \), 0.1m/s, 0.2m/s, 0.5m/s, 1m/s, 2m/s, 5m/s, 8m/s and 10m/s.

3. The boundary condition of the pipe wall is set as no slip condition, and the outlet is set as free outflow.

4. The range of equivalent particle size of the pulp solid phase particles is 100\( \mu m \) ~ 300\( \mu m \), assuming that the solid phase particle size distribution function \( f_N \) satisfies Gaussian distribution, then the average equivalent particle size of solid phase particles \( d_p \) is 150\( \mu m \).

The two-dimensional model and calculation area of pipe are shown in Figure 1. The pipe model is a two-dimensional symmetrical mirror image structure from the pipe axis to the pipe wall. The model consists of the pipe wall, the pipe central axis, the inlet and the outlet. The pipe is axisymmetric, so only half of the radial direction of the two-dimensional pipe needs to be selected as the area for calculation and data analysis. Set the pipe diameter \( D \) as 40\( mm \). In order to ensure that the length of pipe is sufficiently larger than the
length of pipe inlet section $L$, the length of pipe is set as 100D.

In the production process of paper industry, the parameters of pulp fluid transportation system such as temperature and pulp density usually change little, so the calculation area model established in this paper can be assumed as ideal condition. The geometric shape and material parameters are set to be constant, and the influence of temperature and density changes is ignored, so that the robustness of the calculation area model can be guaranteed.

**Computational grids**

The robustness of the fluid model is mainly determined by the number of grids and computer performance. Under the initial conditions and boundary conditions set in this paper, as long as the grid division of the calculation area is appropriate, the model can be guaranteed to have good robustness. If the grid density is reduced, the robustness of the model can be improved to a certain extent, but this will cause the solution to be affected by the number of grids, thus reducing the calculation accuracy and increasing the error.

In order to ensure the grid independence of the solution and to test the robustness of the model, grid testing is necessary. The number of grids tested is respectively the following seven cases: 10,028, 27,672, 48,504, 73,604, 135,562, 303,180, 811,238. In these seven cases, the flow velocity and solid phase volume concentration distribution of pulp fluid with initial average flow velocity $U_{LS} = 1m/s$ and initial solid phase volume concentration $\phi_S = 5\%$ in the fully developed area of the pipe are calculated. In order to ensure that the calculation is in the fully developed area of pipe, the position of pipe away from the inlet 90D is selected, and the radial maximum flow velocity $U_{max}$ and the maximum volume concentration $\phi_{max}$ of this position are shown in Table 1.

Table 1 shows that numerical solutions can be obtained when the calculation area is divided into the above seven grid densities, which indicates that the model has good robustness. When the number of grids divided is 303,180, that is sufficient to obtain the grid independent solutions. Therefore, in this paper the number of computational area division grids is 303,180.

**Numerical simulation and the method for determining length of pipe inlet section of pulp fluid**

The flow velocity and concentration distribution of pulp fluid with initial average flow velocity $U_{LS} = 1m/s$ and initial solid phase volume concentration $\phi_S = 5\%$ in the pipe are numerically simulated.

The flow velocity distribution of pulp fluid near inlet and outlet of pipe is shown in Figure 2.

As shown in Figure 2(a), the flow velocity distribution of pulp fluid in the pipe is constantly changing in the axial direction of the pipe near the inlet, which is caused by the pulp fluid in this section of the pipe is in the pipe inlet section, the flow velocity distribution has not reached the fully developed state. As shown in Figure 2(b), the flow velocity distribution of pulp fluid in the axial direction of the pipe is no longer changed.

![Figure 2. Flow velocity distribution of pulp fluid in pipe ($U_{LS} = 1m/s, \phi_S = 5\%$): (a) near the inlet of pipe, (b) near the outlet of pipe.](image)
near the outlet, this is because the pulp fluid in this section of the pipe is in the pipe fully developed section, the flow velocity distribution reaches the fully developed state and tends to be stable.

Different axial positions of the pipe are selected, and pulp fluid flow velocity distribution data at these positions can be obtained. The flow velocity data at the positions 1D, 5D, 10D, 20D, 30D, 40D, 50D, 60D away from the pipe inlet are respectively selected, and the flow velocity distribution curves of pulp fluid at these positions are shown in Figure 3.

In Figure 3, r represents the distance from any point in the pipe to the center of the pipe, R is the radius of the pipe, \( R = \frac{1}{2} D \). The curve in the figure shows the pulp fluid flow velocity distribution in the pipe at positions 1D, 5D, 10D, 20D, 30D, 40D, 50D, 60D from the pipe inlet respectively. It can be seen that upstream of the 10D position, the flow velocity distribution of pulp fluid in each position in the pipe is obviously different, and downstream of the 10D position, the flow velocity distribution of pulp fluid in each position in the pipeline tends to be stable.

Pearson correlation coefficient method is applied to analyze these curve data, and the relationship between the two sets of flow velocity distribution data at different positions of the pipe can be obtained. The flow velocity distribution data of pulp fluid at any two different positions in the pipe can be represented by sequences
In this paper, the correlation coefficient of flow velocity distribution curves are exactly the same. Therefore, the position where the flow velocity distribution reaches stable state should be found in the area between 10D and 20D. In the pipe area between 10D and 20D from the inlet, select several positions (In this paper, nine positions are selected: 11D, 12D, 13D, 14D, 15D, 16D, 17D, 18D, 19D) and calculate the correlation coefficient between the flow velocity distribution at these positions and the flow velocity distribution at 20D position, as shown in Table 3.

Table 3 shows that in the downstream area of the 13D position, the correlation coefficient $S_Y \geq 99.9\%$ between the flow velocity distribution at each position and the flow velocity distribution at the 20D position. It can be concluded that downstream of the 13D position, the flow velocity distribution of the pulp fluid ($U_{LS} = 1 m/s, \phi_S = 5\%$) reaches the fully developed state.

In order to determine the length of pipe inlet section of pulp fluid, the solid phase concentration distribution of pulp fluid in the pipe should also be studied. The solid phase concentration distribution of pulp fluid ($U_{LS} = 1 m/s, \phi_S = 5\%$) in the pipe is shown in Figure 4.

It can be seen from Figure 4 that in the pipe inlet section, the solid phase concentration distribution of pulp fluid changes obviously, in the pipe fully developed section, the solid phase concentration distribution of pulp fluid tends to be stable.

The solid phase concentration data at the positions 1D, 5D, 10D, 20D, 30D, 40D, 50D, 60D away from the pipe inlet are respectively selected, and the solid phase concentration distribution curves of pulp fluid at these positions are shown in Figure 5.

It can be seen from Figure 5 that the solid phase concentration of pulp fluid near the pipe wall is relatively high, and this phenomenon is more obvious near the pipe inlet. This is due to the position close to the pipe wall is the viscous sublayer, the flow velocity of pulp fluid in the viscous sublayer is close to 0, resulting in the accumulation of solid particles.

Pearson correlation coefficient method is applied to analyze these curve data, and the relationship between the two sets of solid phase concentration distribution

| Table 2. Correlation coefficient of flow velocity distribution ($U_{LS} = 1m/s, \phi_S = 5\%$). |
|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| 1D              | 5D              | 10D             | 20D             | 30D             | 40D             | 50D             | 60D             |                 |
| 1.000           |                 |                 |                 |                 |                 |                 |                 |                 |
| 0.972           | 1.000           |                 |                 |                 |                 |                 |                 |                 |
| 0.938           | 0.990           | 1.000           |                 |                 |                 |                 |                 |                 |
| 0.901           | 0.967           | 0.991           | 1.000           |                 |                 |                 |                 |                 |
| 0.894           | 0.960           | 0.986           | 0.999           | 1.000           |                 |                 |                 |                 |
| 0.900           | 0.964           | 0.988           | 0.999           | 0.999           | 1.000           |                 |                 |                 |
| 0.906           | 0.968           | 0.990           | 0.999           | 0.999           | 0.999           | 1.000           |                 |                 |
| 0.909           | 0.969           | 0.991           | 0.999           | 0.999           | 0.999           | 0.999           | 1.000           | 1.000           |

where $U_{xDi} (i = 1, 2, ...n)$ and $U_{yDi} (i = 1, 2, ...n)$, then the correlation coefficient expression of the flow velocity distribution data of any two sets of pulp fluids is:

$$S_Y = \frac{\sum_{i=1}^{n} (U_{xDi} - \bar{U}_{xDi})(U_{yDi} - \bar{U}_{yDi})}{\sqrt{\sum_{i=1}^{n} (U_{xDi} - \bar{U}_{xDi})^2 \sum_{i=1}^{n} (U_{yDi} - \bar{U}_{yDi})^2}} \quad (13)$$

where $\bar{U}_{xDi}$ and $\bar{U}_{yDi}$ are the mean values of two sets of flow velocity data at different positions in the pipe. The value range of correlation coefficient $S_Y$ is between 0 and 1, that is, $S_Y \leq 1$. Comparing the correlation coefficients of the flow velocity distribution data represented by any two curves in Figure 3. The correlation coefficient $S_Y$ represents the similarity degree of the two flow velocity distribution curves, the greater the value of $S_Y$, the higher the similarity of the two flow velocity distribution curves. If the correlation coefficient $S_Y = 1$ of any two flow velocity distribution curves, then the two flow velocity distribution curves are exactly the same. In this paper, the correlation coefficient $S_Y \geq 99.9\%$ of flow velocity distribution at each downstream position of a certain pipe position is taken as the criterion for the flow velocity distribution to reach stable state. The correlation coefficient of flow velocity distribution curve at each axial position of pipe is shown in Table 2.

Table 2 shows that the correlation coefficient of flow velocity distribution of pulp fluid is above 99.9\% at each position downstream of 20D. In the area between 10D and 20D, the correlation coefficient of flow velocity distribution increased from 99.1\% to 99.9\%. Therefore, the position where the flow velocity distribution reaches stable state should be found in the area between 10D and 20D. In the pipe area between 10D and 20D from the inlet, select several positions (In this paper, nine positions are selected: 11D, 12D, 13D, 14D, 15D, 16D, 17D, 18D, 19D) and calculate the correlation coefficient between the flow velocity distribution at these positions and the flow velocity distribution at 20D position.
data at different positions of the pipe can be obtained. The solid phase concentration distribution data of pulp fluid at any two different positions in the pipe can be represented by sequences $f_{xDi}(i=1,2,\ldots,n)$ and $f_{yDi}(i=1,2,\ldots,n)$, then the correlation coefficient expression of solid phase concentration distribution data of any two sets of pulp fluids is:

$$S_C = \frac{\sum_{i=1}^{n} (\bar{f}_{xiDi} - \bar{f}_{yiDi})(\bar{f}_{xiDi} - \bar{f}_{yiDi})}{\sqrt{\sum_{i=1}^{n} (\bar{f}_{xiDi} - \bar{f}_{yiDi})^2 \sum_{i=1}^{n} (\bar{f}_{xiDi} - \bar{f}_{yiDi})}}$$

(14)

where $\bar{f}_{xiDi}$ and $\bar{f}_{yiDi}$ are the mean values of two sets of solid phase concentration distribution data at different positions in the pipe. In this paper, the correlation coefficient $S_C \geq 99.9\%$ of solid phase concentration distribution at each downstream position of a certain pipe is taken as the criterion for the solid phase concentration distribution to reach stable state. The correlation coefficient of solid phase concentration distribution curve at each axial position of pipe is shown in Table 4.

Table 4 shows that the correlation coefficient of solid phase concentration distribution of pulp fluid is above 99.9% at each position downstream of 10D. In the area between 5D and 10D, the correlation coefficient of solid phase concentration distribution increased from 95.6% to 99.9%. Therefore, the position where the solid phase concentration distribution reaches stable state should be found in the area between 5D and 10D. In the pipe area between 5D and 10D from the inlet, select several positions (In this paper, four positions are selected: 6D, 7D, 8D, 9D) and calculate the correlation coefficient between the solid phase concentration distribution at these positions and the solid phase concentration at 10D position, as shown in Table 5.

Table 5 shows that in the downstream area of the 8D position, the correlation coefficient $S_C \geq 99.9\%$ between the solid phase concentration distribution at each position and the solid phase concentration

---

**Table 4. Correlation coefficient of solid phase concentration distribution ($U_{LS} = 1\,m/s, f_s = 5\%$).**

|   | 1D | 5D | 10D | 20D | 30D | 40D | 50D | 60D |
|---|----|----|-----|-----|-----|-----|-----|-----|
| 1D | 1.000 | 0.771 | 0.672 | 0.655 | 0.632 | 0.622 | 0.603 |       |
| 5D | 1.000 | 0.956 | 0.942 | 0.941 | 0.939 | 0.937 | 0.936 |       |
| 10D| 1.000 | 0.999 | 0.999 | 0.999 | 0.999 | 0.999 | 0.999 |       |
| 20D|       | 1.000 | 0.999 | 0.999 | 0.999 | 0.999 | 0.999 |       |
| 30D|       |       | 0.999 | 0.999 | 0.999 | 0.999 | 0.999 |       |
| 40D|       |       |       | 0.999 | 0.999 | 0.999 | 0.999 |       |
| 50D|       |       |       |       | 0.999 | 0.999 | 0.999 |       |
| 60D|       |       |       |       |       | 1.000 | 1.000 |       |

**Table 5. Correlation coefficient between solid phase concentration distribution at each position in the area between 5D and 10D and solid phase concentration distribution at 10D position.**

|   | 6D | 7D | 8D | 9D |
|---|----|----|----|----|
| 10D| 0.976 | 0.988 | 0.999 | 0.999 |

---

Figure 4. Solid phase concentration distribution of pulp fluid in pipe ($U_{LS} = 1\,m/s, f_s = 5\%$): (a) near the inlet of pipe, (b) near the outlet of pipe.
distribution at the 10D position. It can be concluded that downstream of the 8D position, the solid phase concentration distribution of pulp fluid ($U_{LS} = 1m/s, \phi_s = 5\%$) reaches the fully developed state.

According to the analysis of Table 3, it is known that the flow velocity distribution of the pulp fluid reaches the fully developed state downstream of the 13D position, so the area in which both flow velocity distribution and solid phase concentration distribution reach the fully developed is downstream from the inlet 13D of the pipe. Therefore, it can be determined that the length of pipe inlet section of pulp fluid ($U_{LS} = 1m/s, \phi_s = 5\%$) is 13D.

**Relationship between the length of pipe inlet section of pulp fluid and initial average flow velocity and initial solid phase volume concentration**

The above experiments and analysis give the method to determine the length of pipe inlet section based on CFD under the condition of given specific parameters of pulp fluid. Based on this method, the length of pipe inlet section of pulp fluid under different initial conditions is studied. The initial conditions are as follows: the initial solid volume concentration of pulp fluid is 2%, 5%, 10%, 15%, and 20% respectively; the initial average flow velocity of pulp fluid is 0.05m/s, 0.1m/s, 0.2m/s, 0.5m/s, 1m/s, 2m/s, 5m/s, 8m/s, and 10m/s respectively. The result is shown in Figure 6.

As shown in Figure 6, the relationship between the length of pipe inlet section of pulp fluid and initial average flow velocity and initial solid phase volume concentration is as follows:

1. Under the conditions of different initial average flow velocity and initial solid phase volume concentration, the length of pipe inlet section of pulp fluid is also different;
Under the condition of the same initial average flow velocity, the length of pipe inlet section of pulp fluid decreases with the increase of the initial solid phase volume concentration;

The length of pipe inlet section of pulp fluid always increases first and then decreases with the increase of initial average flow velocity, and when the initial average flow velocity is about 1 m/s, the length of pipe inlet section of pulp fluid reaches the maximum, when the initial average flow velocity increases above 1 m/s, the length of pipe inlet section of pulp fluid decreases continuously with the increase of the initial average flow velocity, and finally tends to be stable.

When the initial average flow velocity is higher than 8 m/s, the difference in the length of pipe inlet section of pulp fluid with different initial solid phase volume concentrations is already very small. When the initial average flow velocity is about 10 m/s, the difference in the length of pipe inlet section of pulp fluid with different initial solid phase volume concentrations is less than 1D.

**Discussion**

The work of this paper has the following problems worthy of discussion:

1. The CFD method applied in this paper is carried out under the ideal condition that the pipe is straight pipe. Some boundary conditions, such as the surface roughness of the pipel wall and the influence of secondary flow of fluid in the pipe, are ignored in the mathematical model. Moreover, environmental factors such as vibration and temperature are not considered. Therefore, the results obtained by this method are approximate and have certain errors compared with the real situation. If the mathematical model is further improved, or combined with a large number of experimental analysis, more accurate conclusions can be obtained.

2. In this paper, Pearson correlation coefficient method is used to analyze the data of flow velocity and solid phase volume concentration of pulp fluid at various positions in the axial direction of the pipe, the more positions are selected in the axial direction of the pipe, the more accurate the results are. Therefore, if the length of pipe inlet section of pulp fluid is determined by the method in this paper, the number of positions selected in the axial direction of the pipe should be as large as possible when the conditions permit.

**Conclusion**

This paper presents a method to determine the length of pipe inlet section of pulp fluid. Based on CFD, the numerical solution of flow state of pulp fluid in the pipe under the given initial conditions and boundary conditions is obtained, and Pearson correlation coefficient method is applied to analyze the flow velocity distribution and solid phase volume concentration distribution data at various axial positions of the pipe to determine the length of the pipe inlet section of pulp fluid. The conclusion are as follows:

1. The length of pipe inlet section of pulp fluid is determined by the initial average velocity and the initial solid phase volume concentration.

2. Under the same initial average flow velocity, the length of pipe inlet section of pulp fluid decreases with the increase of the initial solid phase volume concentration.

3. The length of pipe inlet section of pulp fluid always increases first and then decreases with the increase of initial average flow velocity. When the initial average flow velocity is about 1 m/s, the length of pipe inlet section of pulp fluid reaches the maximum, when the initial average flow velocity increases above 1 m/s, the length of pipe inlet section of pulp fluid decreases continuously with the increase of the initial average flow velocity, and finally tends to be stable.

The work of this paper provides theoretical support for how to determine the installation positions of pulp concentration sensor and flow sensor on the pipe. In addition, the rule that the length of pipe inlet section of pulp fluid is affected by the average flow velocity and concentration under the normal conditions in the paper
industry is summarized, which is of certain significance to the design of the pulp pipe transportation system.

**Declaration of Conflicting Interests**
The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

**Funding**
The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was sponsored by the key scientific research project and Henan province key research and development and promotion special (No. 192102110204) and the national natural science foundation of China under Grant (No. 61805073 & 61801288).

**ORCID iDs**
Song Gao https://orcid.org/0000-0003-2643-3165
Xinwu Du https://orcid.org/0000-0002-8312-5209
Shengjie Jin https://orcid.org/0000-0002-7542-2182
Hao Ma https://orcid.org/0000-0002-4419-5108

**References**
1. Zhou Q. Soft measurement of pulp suspension flow velocity based on wavelet transform. *Can J Chem Eng* 2010; 88(1): 81–87.
2. Qu X, Ikawa M and Shimauchi H. Improvement of the detection of human pulpial blood flow using a laser Doppler flowmeter modified for low flow velocity. *Arch Oral Biol* 2014; 59(2): 199–206.
3. Pithon MM, Sant’Anna LI, Baião FC, et al. Intrusive force and pulp flow measurement. *Am J Orthod Dentofacial Orthop* 2015; 148(6): 891.
4. Ye D, Li H, Zou C, et al. Fluidization characteristics of medium consistency pulp. *J Jiangsu University* 2015; 36(3): 276–280.
5. Wu Y. Integrated measurement of pulp consistency and pulp flow based on sparse decomposition. Dissertation Shaanxi University of Science and Technology, Xi’an, 2016.
6. Mohanty AK and Asthana SBL. Laminar flow in the entrance region of a smooth pipe. *J Fluid Mech Digit Arch* 1979; 90(3): 433–447.
7. Zanoun ES, Durst F and Nagib H. Evaluating the law of the wall in two-dimensional fully developed channel flows. *Phys Fluids* 2004; 16(10): 3509–3510.
8. Lien K, Monty J, Chong MS, et al. The entrance length for fully developed turbulent channel flow. In: *15th Australian fluid mechanics conference*, The University of Sydney, Sydney, Australia13–17 December 2004, pp. 356–363.
9. Launder BE and Spalding DB. The numerical computation of turbulent flows. *Comput Methods Appl Mech Eng* 1974; 3(2): 269–289.
10. Ali L, Islam S, Gul T, et al. Solution of nonlinear problems by a new analytical technique using Daftardar-Gejji and Jafari polynomials. *Adv Mech Eng* 2019; 11(12): 1–10.
11. Gul T, Haleem I, Ullah I, et al. The study of the entropy generation in a thin film flow with variable fluid properties past over a stretching sheet. *Adv Mech Eng* 2018; 10(11): 1–15.
12. Khan W, Idress M, Gul T, et al. Three non-Newtonian fluids flow considering thin film over an unsteady stretching surface with variable fluid properties. *Adv Mech Eng* 2018; 10(10): 1–17.
13. Khan A, Gul T, Zaheer Z, et al. The flow of ferromagnetic nanofluid over an extending surface under the effect of operative prandtl model: a numerical study. *Adv Mech Eng* 2019; 11(12): 1–11.
14. Gul T, Waqas M, Noman W, et al. The carbon-nanotube nanofluid sprayed on an unsteady stretching cylinder together with entropy generation. *Adv Mech Eng* 2019; 11(12): 1–11.
15. Gul T, Noman W, Sohail M, et al. Impact of the marangoni and thermal radiation convection on the graphene-oxide-water-based and ethylene-glycol-based nanoparticles. *Adv Mech Eng* 2019; 11(6): 1–9.
16. Hsu FL, Turian RM and Ma TW. Flow of noncolloidal slurries in pipelines. *AICHE J* 2010; 35(3): 429–442.
17. Lin CX and Ebadian MA. A numerical study of developing slurry flow in the entrance region of a horizontal pipe. *Comput Fluids* 2008; 37(8): 965–974.
18. Ling J, Skudarnov PV, Lin CX, et al. Numerical investigations of liquid-solid slurry flows in a fully developed turbulent flow region. *Int J Numer Methods Heat Fluid Flow* 2003; 24(3): 389–398.
19. Ookawara S, Street D and Ogawa K. Numerical study on development of particle concentration profiles in a curved microchannel. *Chem Eng Sci* 2006; 61(11): 3714–3724.
20. He YR, Chen HS, Ding YL, et al. Solids motion and segregation of binary mixtures in a rotating drum mixer. *Chem Eng Res Design* 2007; 85(7): 963–973.
21. Yakhot V and Orszag SA. Renormalization group analysis of turbulence. I. Basic theory. *J Sci Comput* 1986; 1(1): 3–51.
22. Xiaobing L and Liangjun C. A K-ε two-equation turbulence model for the solid-liquid two-phase flows. *Appl Math Mech* 1996; 17(6): 523–531.
23. Gidaspow D. Multiphase flow and fluidization: Continuum and kinetic theory descriptions. New York: Academic Press, 1994.
24. Cheng TS and Yang WJ. Numerical simulation of three-dimensional turbulent separated and reattaching flows using a modified turbulence model. *Comput Fluids* 2008; 37(3): 194–206.
25. Barbhuiya AK and Talukdar S. Scour and three dimensional turbulent flow fields measured by ADV at a 90° horizontal forced bend in a rectangular channel. *Flow Meas Instrum* 2010; 21(3): 312–321.
26. Yahao S, Zhengqi G, Shuichang L, et al. A study on the applicability of k-ε model for numerical simulation of automobile external flow field. *J Hunan University Technol* 2019; 33(1): 66–72.
27. COMSOL Multiphysics®cn.comsol.com. Stockholm, Sweden: COMSOL AB.