CFX Simulation of Slurry Pipeline Flow

Xiong Ting
Wuhan University of technology, Wuhan, Hubei Province, China
Tel.: 13237181778
E-mail: bearmos@163.com

Received: 9 October 2013 / Accepted: 22 November 2013 / Published: 30 December 2013

Abstract: This paper explores how to have a numerical simulation for the slurry transportation process by use of CFD. Euler’s method and Lagrange method are applied respectively to calculate pipeline pressure distribution and loss. After the simulation of slurry settling process in pipes, its numerical results get verified in the tests. The results demonstrate that Euler model simulation comes closest to actual working conditions in the case of high-concentration slurry as the transmission medium, while Lagrange model comes closest to actual working conditions in the case of coarse particles or sandy soil. Copyright © 2013 IFSA.

Keywords: Slurry flow, CFX, Simulation, Eular model, Lagrange model.

1. Introduction

Slurry pipeline transport is widely applied in projects such as dredging, hydraulic reclamation, mining and transportation [1]. Researches on slurry pipeline flow are helpful to the improvement of efficiency and safety and to the reduction of silting and costs in these projects by offering key technical parameters. Slurry pipeline transport is a solid-liquid two-phase flow. As for it, worldwide professionals and experts have made different kinds of researches, including theoretical research, experimental study and numerical simulation [2, 3]. In theoretical research, physical model is built to get corresponding flow equations which are solved by mathematical analysis. Usually these solution results are precise and therefore have wide applicability. As for the flow which cannot be theoretically analyzed, the preciseness of solution is usually greatly impaired although theoretical models can be built by means of simplification. Therefore, other experimental methods are needed to be adopted. These experiments usually have theoretical guidance and related simulation tests. Through induction and analysis these experimental data can provide some empirical equations and relation charts so as to help the real projects. But the limitation of experiment conditions makes the experimental conclusions not widely applicable. With the rapid development of computer science, many complicated equations are gradually taking numerical solutions. This paper also adopts CFD to calculate and analyze the slurry pipeline transport [4].

2. Theory Model

Due to the different motion forms of solid particles, pipeline slurry transport has three following basic kinds: transport of fake homogeneous flow, transport of clean-water two-phase flow, and transport of seriflux two-phase flow.

2.1. Transport of Fake Homogeneous Flow

When the solid particles are small, they will become homogeneous seriflux after mixed with
water. These particles will not change and settle in the process of transportation. The pipeline transportation resistance follows pipeline transportation resistance laws of the non-Newtonian fluid. The unit weight and viscosity of seriflux changes with its concentration and accordingly influences the pipeline resistance. Such case can adopt Bingham fluid model [5]. The velocity of Bingham fluid should follow the equation (1):

\[
\frac{\partial^2 (\eta u)}{\partial x^2} + \frac{\partial^2 (\eta u)}{\partial y^2} = p
\]

where \( \eta \) is the apparent viscosity of Bingham fluid, Pa·s; \( \mu \) is the viscosity of Bingham fluid, Pa·s; \( \tau_0 \) is the yield stress of Bingham fluid, Pa; \( p \) is the unit pressure drop, Pa/m.

As for laminar flow, the rheological equation is:

\[
\tau = \tau_B + \eta \frac{du}{dy},
\]

where \( \tau_B \) is the shear stress of Bingham fluid.

### 2.2. Transport of Clean-Water Two-Phase Flow

Under such circumstance, solid particles are large and coarse with few small ones. The particles will obviously settle and present uneven concentration distribution in pipeline fracture surface [6]. The particles which are pushed forward take up a large percentage. When flow velocity is low, the slurry concentration of upper pipes is low and even near to clean water. Flow velocity distribution is calculated respectively from the aspects of clean-water turbulence and sandy flow.

\[
\frac{u_{\text{max}} - u}{U^*} = \frac{1}{\kappa} \ln \frac{y_m}{y}, \quad \Delta \leq y \leq y_m,
\]

where \( y \) is the height from the bottom; \( y_m \) is the height of the maximum velocity; \( u, u_m \) is the velocity; \( \kappa \) is the Carmen constant; \( U^* \) is the friction velocity; \( \Delta \) is the adhesive layer thickness.

As for sandy flow area, the velocity distribution has already been tested by H. A. Einstein, Ning Cheien, V. A. Vanoni, G. N. Nomicos and al, and gets the following conclusions: flow velocity in main flow area accords with the logarithmic distribution, but Carmen constant decreases with the increase of sandy concentration or vertical concentration gradient. The turbulence of sandy flow has a more uneven velocity distribution than water clean water, as is shown in that: the flow velocity gradient in main flow area is larger than that of clean water turbulence, while the flow velocity gradient in near base area is close to that of clean water flow.

Because the suspending solid particles in pipes are low in concentration, pipeline resistance in such flow can be approximately assumed as turbulence loss of clean water flow plus traction loss of solid particles. In the scope of real flow velocities, usually a simplified equation is used to analyze the pipeline pressure by adding turbulence loss of clean water to extra loss caused by solid particles. That is

\[
J = J_0 + \Delta J,
\]

where \( J_0 \) is the Clear water hydraulic gradient, \( \Delta J \) is the extraneous loss.

### 2.3. Transport of Seriflux Two-Phase Flow

Under such circumstance, a certain amount of small particles and water will mix up to be seriflux, which reduces the settling speed of the coarse particles or turns their throwing motion into floating motion. When the small particles in the suspending liquid exceed a certain amount, they will form flocule net organization and then reduce greatly the settling velocity because of the net resistance. Therefore there goes a roughly homogeneous vertical distribution of sand content. Under such flow velocity, most solid particles maintain floating motion, and only a few larger particles are in the transition stage between floating motion and throwing motion. Hence, the pipeline resistance reduces a lot.

Due to the decrease of settling velocity and density gradient of vertical distribution, here reduces the turbulence energy consumption for supporting slurry weight and overcoming density gradient, and goes a homogenizing flow velocity distribution. The tests verify that the flow velocity distribution in main area still follows logarithm law.

There are several kinds of methods in view of resistance calculation. When designing the coal slurry pipelines, Wasp et al proposed to divide the solid particles into two kinds – evenly floating ones and base-gliding ones, and then add these two resistances. The resistance of evenly floating solid particles is calculated as homogeneous slurry resistance, while the resistance of base-gliding solid particles is
calculated by Durand formula. Others also put forward to follow the calculation of clean water as liquid phase, only replacing the clean water turbulence resistance with slurry turbulence resistance. The extra resistance is caused by larger particles.

The pertinent literature [7, 8] points out that the difficulty of predicting pipeline resistance for slurry as the liquid-phase lies in how to clarify the percentage of slurries in throwing motion under high-concentration slurry condition. Therefore, if there is a method which can clarify the respective percentages of particles in throwing motion and particles in floating motion under different flow velocities, the corresponding pipeline resistances under different flow velocities will be easy to find out.

3. Simulation Calculation

Slurry pipeline flow can be calculated by three different models, which are transport model of fake homogeneous flow, transport model of clean-water two-phase flow, and transport model of slurry two-phase flow. This paper applies CFX software to simulate the pipeline slurry flow. In order to simplify related calculations, pipelines are set as horizontal ones (models and grid distribution are shown in Fig. 1), and adopts the first two models for simulation calculation. In CFX, the first model can choose Euler’s method for calculation, and the second model can choose Lagrange method for calculation [9].

3.1. Euler Method

With Euler’s method for simulation, slurry can be an individual phase. In CFX slurry is set as a substance and given related parameters. Starting conditions are input, shown as Fig. 2.

Euler’s Method has the following features: each phase are mixed in a macro-scale, which is far less than analysis scale (grid scale), but much more than molecule scale; all phases occupy a same space volume and in controlled volume each phase’s volume can be present by variable volume fraction; each phase has its own flow field; each phase has its coupling by models of energy transfer, momentum transfer and quality transfer. Most Euler’s models are based on empirical formula. According to this model, internal pipeline pressure distribution is shown as Fig. 3, and pipeline pressure loss is shown as Fig. 4.

The above analyses reach the conclusion: in Euler’s method simulation, pressure distribution and pipeline pressure loss of slurry are close to those of clean water as slurry is treated as homogeneous flow and calculation is simplified as one-phase one.
3.2. Lagrange Method

Lagrange Method has the following features: it tails after the representative particle sample to go through continuous flow; as for each particle its location and velocity are acquired by ordinary differential equation; the total quality is separately given to representative particle samples and therefore each particle has its own quality flow rate and its own numerical flow rate. In Lagrange model, slurry particle is treated as an individual phase and water is seen as another phase. Analysis can be made by use of particle-tracing model in CFX. The specific setting is shown in Fig. 5.

According to this model, pipeline pressure distribution is shown in Fig. 6, slurry particle velocity is shown in Fig. 7, pipeline pressure loss is shown in Fig. 8, and the particle volume fraction in pipe base is shown in Fig. 9.
The above analyses show that: the velocity of slurry particle is increasing first and then decreasing from up to down along the pipe diameter (direction of axis Y) as slurry particle and clean water are two separate phases in Lagrange model (Fig. 7). Such increase appears because of that upper slurry particles are reducing in number by settling and large quantities of slurry particles in the middle pipes accelerate by the drive of pump. Such decrease appears because of that slurry particles settle at the bottom of the pipes and then increase the pipe friction. As shown in Fig. 9, as conveying distance extends, volume fractions of bottom slurry particles are rising. It can also illustrate that slurries have their settling in the process of movement.

4. Experimental Verification

In order to verify the accuracy of calculation theory of CFD, the calculation results are compared with the laboratory test results of the mud. The test conditions are introduced briefly as follows:

The whole test system includes pool, slurry pipe (diameter 50 mm), glass tube, slurry pump, and motor and control valve and so on. Its main task is to transport slurry. The test bed is a closed cycle system, which means that the slurry sucking from the slurry pool still get sucked back to the slurry pool. The experimental medium of the test bed is a solid-liquid mixture, which is likely to separate and settle in the stationary state. Therefore, in order to keep slurry a uniform concentration in the experimental process, a stirring pump is specially equipped to mix slurry pump and keep the liquid of the pool a suspension state. Two transparent glass tubes are installed to observe the multi-phase flow in the slurry pipe. The physical map is as follows:

From the large experimental data, this simulation chooses the representative working conditions for comparison: the slurry concentration is 40 %, and the rate of flow is: 1.45 m/s. as for evaluation index it chooses the common pipeline pressure drop to evaluate slurry pipeline flow resistance.

When slurry in the experiment flows with the speed of 1.45 m/s, the monitoring sections of pressure gauge 4 and 5 are treated as the experimental sections. With the monitoring section of pressure gauge pipeline 4 as the entrance, and the monitoring section of pressure gauge pipeline 5 as the exit, it obtains the numerical values of two pressure sensors 5 meters between. That is, the entrance and exit pressure value. Records of the experiment are as follows: the pressure Table 4 shows that the pressure value is 6.87 kPa; the pressure Table 5 shows that the pressure value is 4 kPa. Because the experimental section is five-meters-long, for comparison with the simulation results of the demand, the values have to be converted to an average pressure drop within a meter for the sake of comparison with simulation results. Hence, the average pressure drop of entrance and exit pressure value is calculated to be 0.57 kPa, namely 570 Pa.

From the experimental data, the errors of Euler’s method and Lagrange method are calculated by being compared with actual flow. The calculation list is as follows (Table 1):

By comparison of the results of CFD simulation and experimental data, it can be learned that Euler’s method is more close to the experimental result. Such phenomenon is caused by the following reasons:
1) The slurry medium of the test belongs to clay, which turned into homogeneous slurry after mixing with water.

2) The test set the slurry flow rate as 1.45 m/s. According to the formula calculation it is a turbulent flow, which has little sedimentation, high concentration up to 40 %, and pipeline slurry flow near the fake homogeneous flow.

<table>
<thead>
<tr>
<th>Simulation pressure drop (Pa)</th>
<th>Practical pressure drop (Pa)</th>
<th>Numerical error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Euler method</td>
<td>650</td>
<td>570</td>
</tr>
<tr>
<td>Lagrange method</td>
<td>1350</td>
<td>570</td>
</tr>
</tbody>
</table>

### Table 1. Error analysis.

#### 5. Conclusion

In summary, according to different working conditions the slurry pipeline flow can be treated as transport of fake homogeneous flow, transport of two-phase flow between slurry particle and clean water, and transport of two-phase flow between slurry particle and slurry. Euler model simulation comes closest to actual working conditions in the case of high-concentration slurry as the transmission medium. Judging from the theoretical model features, Lagrange model comes closest to actual working conditions in the case of coarse particles or sandy soil, because slurry has an evident settling process.

### References

[7]. Qian Ning, The hyperconcentrated sediment flow, Qinghua University Press, Beijing, 1989.