Simulation of the Flow Field of Cement Mixer Based on Numerical Methods

Li Shuiping, Li Xiaotian, Shi Lugang
School of Mechanical and Electronical Engineering, Wuhan University of Technology, Wuhan 430070, P.R.China

Abstract *Different kinds of research methods of the flow field in the cement mixer were introduced and compared, theory bases of the numerical simulation method of the flow field in the cement mixer were described, whole of thoughts of realization this method were analyzed. And then, the geometrical model and mathematical mode of the cement mixer were created by Fluent, which was based on CFD theory; and the characteristic of flow field in the cement mixer was simulated by Mixture model, SIMPLE algorithm and standard k-ε turbulence model. The calculation results showed that the distribution of flow field in cement mixer could be simulated preferably by the numerical technology, which supplied the valuable theoretical guidance for designing of the cement mixer.*

Keywords *Cement mixer Numerical simulation Flow field*

1.Introduction

Cement slurry is frequently needed to grout the cracks of the buildings when reinforcing the constructions. Currently, most cement slurry is produced by the common mechanical mixer which makes grout by vane, scraper or spiral device. And the mixer is composed by gear, mixing shaft and impeller. It will easily produce the dead angle and precipitation at the bottom of mixing tank in the mixing process due to the blade is fixed on the shaft. Apart from that, it has the problems which are big energy waste, low efficiency and can not prepare for different water-cement ratio cement slurry fast. Therefore, the development of a new type of high-performance cement slurry mixing equipment has become rather important.

Currently,cement mixer commonly can be divided into three categories:colloidal mills mixer, blades mixer, water-jet mixer. Although these devices are generally able to achieve the desired technical effect, the mixing efficiency and cost is another matter. colloidal mills mixer is an advanced equipment, mixing, high efficiency, high concentration slurry can be stirred, but the structure is more complex and costly. the structure of blades mixer is simple, the cost is inexpensive, and widely used in engineering, but the efficiency is low. water-jet mixer is a low-cost equipment, have the characteristics of high speed stirring, but because the mixing uniformity is not high, to some extent these limit its wider use. So how to provide advanced cement mixing equipment is an important issue that engineering and technical personnel in the country have to faced.

For the design of cement mixer, traditional theory and experimental measurements usually used in analysis of the flow field. Study found that in the mixer there are various of slurry flow forms at the same time, combined with the interaction of cement powder and water, make the two-phase flow field very complex and difficult to carry out a detailed theoretical analysis. Also the complex flow characteristics and opacity of the slurry limit the application of experimental methods. In process of today's research about fluid mechanical design, the CFD method to simulate the flow fluid inside the machine has become an important technical means when the software of CFD have emerged. In engineering, numerical simulation has been more widely

used, it can be replaced experiment to cut down the cost in some extent, In addition, numerical simulation can provide a lot of information about flow field, it can give reliable basis for the optimal design of fluid machinery for designers.

Comprehending of the cement slurry flow characteristics and the mechanism of mixing is a basic prerequisite for the structural design [1]. The main purpose of this paper is to simulate and analyze the flow field of the equipment based on the numerical simulation, and then study the mixed performance.

2.Research Methods of the Mixer Flow

2.1 Introduction of Research Methods of the Flow Field

2.1.1 Theoretical analysis method

On the basis of the relevant experimental data and the law of conservation of energy and so on, it will establish the velocity field distribution of fluid flow, the stress field distribution of fluid flow , mass conservation law and energy conversion law by the mathematical analysis, thus construct the mathematical model of fluid flow[2] .As a result, it can forecast the intensity and characteristics of the fluid flow of mixer as well as its function and influence on mass transfer and energy exchange.

2.1.2 Experimental Methods

(1) Visualization of fluid method

Visualization of fluid method which can monitor the trajectory of fluid is a basic method of studying flow field. There are anisotropic thin (particulate) method, laser-induced fluorescence method and dye tracer method [3].

Anisotropic thin (particulate) act refers to adding the anisotropic particles (such as titanium dioxide, mica, aluminum stearate, etc.) which have the reflective capacity to the fluid so that the anisotropic particles flow with the fluid, and the strong reflective capacity of particulate makes the flow trajectory of fluid can be visible. In addition, we are able to study the movement law of two-phase fluid and the migration law of material according to the selective dispersion of some particles in fluid (for example, water phase is chosen for mica, while hydrophobic organic phase are chosen for aluminum and mica).

(2) Video measurement

Using advanced camera equipments (such as magnetic resonance fluid imager, Laser Doppler veloeimetry, particle image vefocimetry etc.) to film the track of fluid flow and then obtain it's boundary conditions as well as the related parameters [3]. PIV technique is non-contact measurement technology which has many advantages such as high measurement accuracy, wide range of speed measurement speed, having simple principle and small influence from outside .Consequently, it is able to depict the whole velocity vector of the total velocity field, so the flow field information of one profile can be analyzed at once.

2.1.3 Numerical Simulation Method

Numerical simulation is that adopting numerical method and logical way to reflect changes prototype process and the rules of movement in the computer. Mathematical model, different from the physical model, is often made from a single equation, a group of equation or groups formulas. We can get the various locations of the basic physical quantities (such as speed, pressure, temperature, concentration, etc.) and their changes in the very complex flow by numerical simulation. In addition, it can optimize the design of structural by combining with CAD.

2.2Comparison of the Research Methods of the Mixer's Internal Flow Field

Table 1 Comparison of the Research Methods

	Fheoretical Nnalysis Method	Experimental Method	Numerical Simulation
Main Advantage	The results whose all kinds of influencing factors are clearly visible have universality. It is the theoretical basis which is used to guide the experimental research and verify the new numerical calculation method.	The experimental results which are true and believable,are the basis of theoretical analysis and numerical method.	It is easy to choose different physical parameters to make the various effective and sensitive tests because you are not restricted by physical model and mathematical model.
Main Disadvantages	We are often required to abstract and simplify the object of calculation. For the non-linear situation, only the analytic results of simple flows can be given.	Experiments are often limited by model size, flow disturbances and measuring accuracy, in addition ,they will also encountered lots of problem such as the enormous cost of funding and manpower,material resources and the long cycle.	It strongly rely on the mathematical model,and numerical processing method will lead to false results.

Numerical simulation method combined with traditional theoretical analysis method and experimental measurement method form a perfect system of studying the problems of the mixing equipment's internal fluid flow.

From table 1, we can we see clearly that these methods have their own advantages and disadvantages. Both traditional theoretical analysis method and experimental test method can not obtain the characteristics of flow field mainly due to the following two points: first, the visualization methods such as staining PIV and tracer method can not be used to measure the flow field which has high concentration and poor visibility; second, at present, all of the contact velocimeter can not test the velocity vector of cementing slurry flow field accurately. Therefore, the slurry flow field can only be studied by numerical simulation.

3. Numerical Simulation Method of the Internal Flow Field of Mixer

3.1 Theoretical Basis of Numerical Simulation

The theoretical basis of numerical simulation is computational fluid dynamics. Fluid flow is dominated by physical conservation laws which relates to the principal basic laws including mass conservation law, momentum conservation law, energy conversion and conservation law, components conversion and balance law，and so on[2,5]. These laws of the basic equations can be expressed as follows:

$$\frac{\partial(\rho\phi)}{\partial t}+div(\rho\bar{u}\phi)=div(\Gamma_{\phi}grad\phi)+S \tag{1}$$

Expand the form of:

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u\phi)}{\partial x}+\frac{\partial(\rho v\phi)}{\partial y}+\frac{\partial(\rho w\phi)}{\partial z}$$
$$=\frac{\partial}{\partial x}(\Gamma\frac{\partial\phi}{\partial x})+\frac{\partial}{\partial y}(\Gamma\frac{\partial\phi}{\partial y})+\frac{\partial}{\partial z}(\Gamma\frac{\partial\phi}{\partial z})+S \tag{2}$$

Where ϕ is the general dependent variable, and so can represent the solution of variables of u、v、w、T; Γ is the generalized diffusion coefficient; S is the generalized source term.

In the macro category of macro continuous mechanics, two-phase flow analysis can be divided into two types: (1) local homogeneous flow analysis; (2) two-phase slip flow analysis. Cementing slurry flow in stirred tank belongs to two-phase slip flow which mainly has three types of analysis model: Euler-Lagrange model, Eulerian-Eulerian model that is a mixture of the two-fluid model and Mixture model. Euler-Lagrange model centralizes the advantages of macroscopic pseudo-fluid model and micro-dynamics model, that is, take into account not only the solid phase and liquid-phase interactions, and can describe the collision between particles, but it needs to record the spatial location and movement of every particle in the flow field , therefore a larger computer memory is necessary. Currently, there are certain difficulties in the simulation of high concentration of solid-liquid two-phase flow [4, 5]. At present, multi-fluid model is given a wide range of applications. In a multi-fluid model, the continuous (liquid) phase movements is described by Euler equations, assuming that the dispersed (particle) phase is the mutual penetration and continuous quasi-fluid, its movement is also described by Euler equation, and then solve the momentum, energy and quality equations of each phase. However, the computational complexity of multi-fluid model results in poor stability, so it confronts many difficulties in the practical application.

Mixture model is a simplified multiphase flow model, which is used to simulate each phase of the multiphase flow at different speeds, however, the local equilibrium under short spatial scale is assumed. For mixture model, there is a strong coupling between different phases. When there exist a wide range of particle distribution, Mixture model can achieve good results as perfect multiphase flow model. Due to space limitations, the control equations of two-phase flow will not be listed here.

3.2 The Process of Numerical Simulation of the Internal Flow Field

At present, the numerical simulation of mixing process is on the basis of macro flow field. First of all, it obtains the velocity field distribution law of macro-flow field and then joins tracer when the flow field is stable. Through the calculation of tracer concentration changes with space and time to simulate the mixing process. This method applies to the flow field of relatively low concentration. In spite of that, there are great difficulties in the simulation of high concentration slurry flow field. In this paper, cementing slurry two-phase flow field was simulated by mixture model, and then the mixing effects were assessed according to the results of simulation. The aim of numerical simulation is to obtain the useful data used to instruct the optimal design of mixer, such as mixing time and mixing power number. The basic flow of numerical simulation is shown in Figure1.

In the process of numerical simulation, we can create the geometric model according to the structure of mixer. Generally, we should deal with the physical mode reasonably during the establishment of geometric model since because it is usually rather complex [6, 8]. In order to reduce the unnecessary computation time, the physical model need to be simplified as much as possible, such as the cover of the mixer which has little impact on the effect of stirring, therefore, it ought to be neglected during the geometric model establishment; for those structures which have greater impact on the effect of stirring, not only to retain but also to establish them accurately. The establishment and dispersion of the mathematical model which directly affects the accuracy and precision of simulation are the core parts of numerical analysis. The key of numerical simulation is to establish the mathematical model which can reflect the essence of engineering problems. For different fluids, we can choose different suitable mathematical models, such as RNG k-ε two-equation turbulence model or two-Realizable k-ε model may be the best choice in a flow field with a cyclone or strong bending wall [9]. We discretize the regional of calculation before numerical calculation, that is, divide the space of calculation into many sub-regions, and to identify nodes in each region in order to generate grid. And then, the control equations are discretize in the discrete grids, as a result, partial differential equations turn into the algebraic equations of each node for numerical solution. Algebraic equations have a

variety of discrete formats including QUICK format which can provide higher calculation accuracy when we study swirling flow by structured grid. So the QUICK format may be the best choice during the numerical simulation of the internal flow field in mixer.

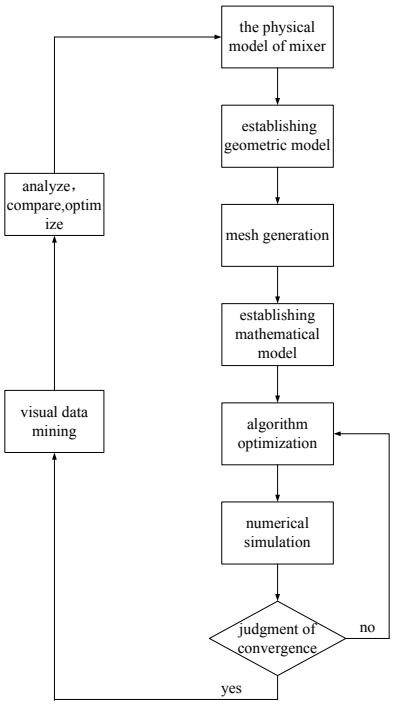

Figure 1 The basic flow of numerical simulation

Apart from a few simple questions, the discrete equations can not be directly solved, because certain adjustments of the dispersion equations are necessary, and we ought to deal with the order and way of unknown variables (speed, pressure, temperature, etc.) especially in order to solve them easily. SIMPLE algorithm is widely used in engineering. Before carrying out the computer simulation, setting the model boundary conditions is required because the boundary conditions which include the inlet velocity, boundary conditions, outlet pressure, the concentration of slurry, are necessary for solving the discretized equations. Setting appropriate initial value and sub-relaxation factor appropriate before the iterative solution in favor of iterative calculations converge faster and more easily. Once convergence of iterative solution achieves prescriptive convergence precision, the data we need can be obtained through the user interface.

4. Process of Numerical Simulation of Flow Field of Mixer by Fluent

4.1 Tools of Numerical Simulation of Flow Field

On the basis of the theory and method above, we can achieve the numerical simulation of flow field by Gambit and Fluent soft wares [10]. Fluent is specific CFD software which is used to simulate and analyze a complex geometry in the field of fluid flow and heat exchange while GAMBIT is professional pre-processing software for Computational Fluid Dynamics, its main features include three aspects: geometric modeling, mesh generation and assignment of boundaries. Among them, mesh generation is the most important function. It generates grid file containing boundary information. Fluent can realize numerical simulation of flow field of mixer effectively in connection with its pre-processing software Gambit.

4.2 The Establishment of Geometric Model

Mesh generation is the necessary condition to establish discrete control equations, because the quality of the mesh has a direct impact on the accuracy of simulation results. Through analyzing the physical model of mixer, this paper used Gambit to establish the mesh model on the basis of geometric model of mixer, as shown in Figure 2.

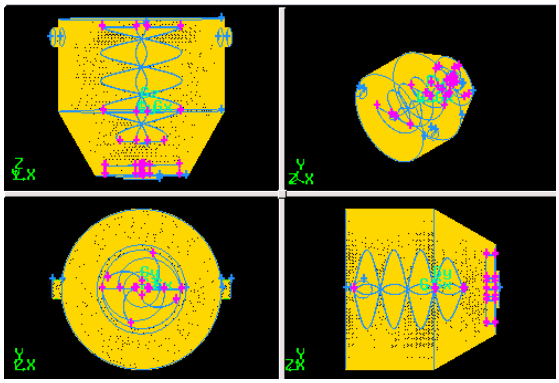

Figure 2 The mesh model of mixer

To reduce quantity of mesh cells and to improve precision and convergence, we used structured mesh in the nozzles. For the internal structure with complex geometric shape, the flow field changes rapidly, it is inconvenient to adopt structured mesh, so a structured-unstructured mixed type mesh was employed in this computation.

4.3 Solve the Discrete Mathematical Model

Mesh model file will be optimized after being imported in FLUENT. We discreted control equations of mesh nodes by numerical discretization of QUICK format in this paper. Numerical algorithm is one of the methods for solving the discrete control equations of grid model, and SIMPLE algorithm is quite applicable.

4.4 Preliminary Analysis of Simulation Results

To a large extent, the effect of mixing depends on the internal structure of flow field, so the existence of turbulence which has complex flow field is an important prerequisite for enhancing the effect of mixing.

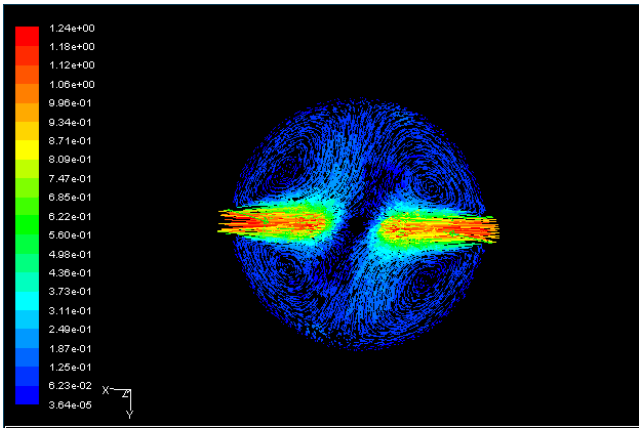

Figure 3 Velocity vectors colored by radial velocity (m/s)

After analyzing the velocity vector distribution chart, it can be seen that the mixer produced a strong vortex under the action of the spiral blade combined the nozzles. The radial velocity is

shown in figure 3.

5. Future Research

At present, the literatures about the numerical simulation of cement mixer are rare. Although the numerical simulation method has its unique advantages, there are many aspects need to be further improved, which mainly include the numerical simulation accuracy and convergence rate.

Numerical simulation accuracy of multiphase flow mainly depends on the discrete mathematical model, therefore how to establish mathematical model in accordance with the mechanical structure of mixer is one of the difficulties of numerical simulation. Low–order dispersion and high-order dispersion has its own excellent shortcomings, so the way to solve the discrete control equations is a very key technology. The purpose of numerical simulation of flow field is obtaining the useful data to optimize the mechanical structure, thus making further analysis of the simulation results is one of the future research directions.

Acknowledgements

This research is funded by the National Key Technology R&D Program of China (No. 2006BAJ03A09-08).

References

[1] Zhao Jianjun，Yuan Shouqi，Liu Houlin, Huang Zhongfu and Tan Mingao. Simulation of solid-liquid two-phase turbulent flow in double-channel pump based on Mixture model. Journal of Transactions of the CSAE, 24 (1)(2008) 7-10.

[2] Wang Fujun. Analysis of Computational Fluid Dynamics—Principle and Application of CFD Software. Beijing: Tsinghua University Press , 2006.

[3] Zhan Hanhui, Chenghao, Liu Jianwen and Zhan Xuehui. Principle of Secondary Flow. Changsha:Central South University Press, 2005.

[4] Anders Darelius, Anders Rasmuson, Berend vanWachem, Ingela Niklasson and Staffan Folestad. CFD simulation of the high shear mixing process using kinetic theory of granular flowand frictional stress models. Journal of Chemical Engineering Science, 63(8)(2008) 2188-2197.

[5] Zhou Lixing. Dynamics of Multiphase Turbulent Reacting Fluid Flows, Beijing: National Defense Industry Press, 2002.

[6] Wu Bo,Yan Hongzhi,Duan Yiqun. Study on 3-D Turbulence Numerical Simulation and Wear Characteristics of Slurry Pump. Journal of China Mechanical Engineering. 20(6)(2009) 719-780.

[7] Huang Si, Wang Guo-yu. A 3D Numerical Simulation of Solid-liquid Turbulent Flow with High Solid-concentration in a Centrifugal Pump. Journal of Coal Mining Machinery. Vol.11(2005) 53-57.

[8] K.H. Javed, T. Mahmud and J.M. Zhu. Numerical simulation of turbulent batch mixing in a vessel agitated by a Rushton turbine. Journal of Chemical Engineering and Processing, 45(2)(2006) 99–112.

[9] Benjamin Coesnon, Mourad Heniche, Christophe Devals, Franc¸ois Bertrand and Philippe A. Tanguy. A fast and robust fictitious domain method for modeling viscous flows in complex mixers: The example of Propellant make-down. International Journal for Numerical Methods In Fluids, 58(4)(2008) 427–449.

[10] Fluent Inc. FLUENT User Guide.2003.