Advances in Engineering Research, volume 149
2nd International Conference on Mechanical, Electronic, Control and Automation Engineering (MECAE 2018)

Numerical Simulation and Analysis Optimization of Cast-In-Place Process of Non-Bearing Wall Based on CFD

Jiabin Ma [a, *], Weiguang Xiong, Luhang Zhao, Jiajie Wang, Shihang Wang
Northeastern University at Qin huangdao, Qinhuangdao Hebei, 066004, China
[a] m18716038108@163.com

Keywords: Wall Pouring, CFD, Flow Field Simulation, Optimal Design, ANSYS Fluent.

Abstract. This article takes the non-bearing wall cast-in-place process as the research object, making comprehensive use of CFD analysis methods, CAE technology, ANSYS Gambit and Fluent software. the flow field simulation was carried out in the mixing section and the grouting process in the pouring wall process. Finally, the flow field simulation was carried out on the stirring section and the grouting section in the process of pouring the wall. At the same time, the velocity field and pressure field in each time period were analyzed, the influence of different wall materials on each link was studied and the optimized cast-in-place process parameters were obtained.

1. Non-Bearing Wall Construction Status and Process Analysis

1.1 Existing Wall Construction Methods

At present, for the construction of non-load-bearing walls, the main wall materials are fly ash aerated bricks, ceramsite blocks, gypsum porous lightweight bricks and partition boards. The construction method of such a wall needs to be completed manually or by slurry wall materials. It is necessary to manually complete the installation and support of a plurality of vertically arranged templates of a wall and then inject the entire wall at one time. The traditional construction method has many deficiencies. In particular, the process of pouring the slurry is often faced with disadvantages such as insufficient wall strength and flatness, etc.[1]-[3].

1.2 Process Analysis of Wall Pouring

With the rapid development of China's construction industry, the amount of wall construction has been increasing year by year. More and more non-bearing walls have been selected for construction by pouring the entire wall or by means of sub-module pouring. For the cast-in-place process, the selection of raw materials for the slurry, selection of the agitation method, setting of the grouting pressure, etc., will directly or indirectly affect the quality of the pouring wall. For the traditional adjustment of optimization parameters based on human experience, a lot of manpower, material and financial resources are wasted [4]-[6].

With the continuous development and progress of computer technology, Computational Fluid Dynamics (CFD) has gradually emerged in the field of fluid mechanics research. It quantitatively describes the numerical solution of the flow field in time and space through computer numerical calculation and image display methods. To achieve the purpose of researching practical problems. Fluent, as a common CFD software package in the world, has a wealth of physical models and solving methods, can accurately simulate a variety of complex fluid problems, and has a high degree of credibility. Therefore, the introduction of CFD research method combined with Fluent software not only can simulate the complex fluid field, velocity field and pressure field in the process of stirring and grouting, but also can save a lot of resources and reduce the production cycle [7]-[10] .

2. Flow Field Simulation Analysis Under ANSYS Fluent Environment

2.1 Selection of Materials

This paper mainly analyzes cast-in-place gypsum slurry as the main material for industrial use of fly ash, phosphogypsum, desulphurized gypsum and other industrial by-products of gypsum chemical waste, which can achieve environmental protection and reduce costs. Through experiments, it can be proved that the use of such materials can guarantee the quality of the wall to meet the standard, combined with the new non-load-bearing wall in-situ pouring equipment, can significantly reduce the construction period, improve the construction speed. Table 1. Physical parameters of the slurry used for this simulation.

Table 1. Properties of cast-in-place gypsum slurry

Material	Density	Specific-Heat	Thermal Conductivity	Viscosity
FGD-Liquid	1006.2kg/m^3	1100j/kg-k	0.312w/m-k	1.1kg/m-s

2.2 Flow Field Simulation in A Stirred Environment

This paper mainly simulates the flow field of the rotation of the double impeller in the process of stirring. The calculation region is 3m long and 2m wide. There are two rotating impellers in the middle and rotate clockwise. The two rotating impellers are separated by 0.5m, as shown in Figure 1.

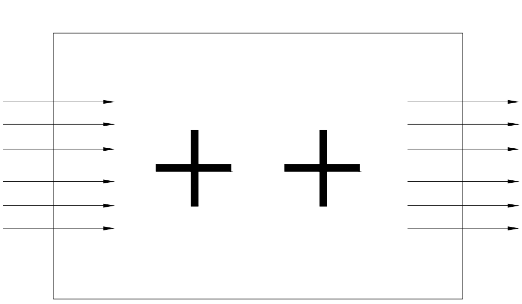


Figure 1. Double impeller rotation model

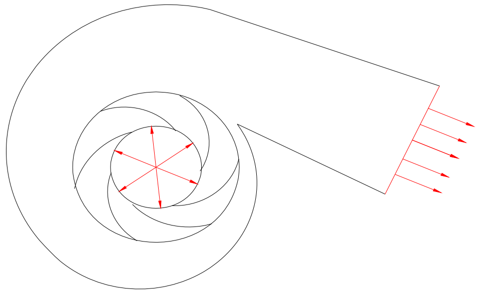


Figure 2. Centrifugal pump two-dimensional model

Meshing is done by Gambit and the grid information is obtained. There are 198714 nodes, 65743 grid surfaces, and 46871 grid cells. The inspection of the grid by Fluent shows that neither the minimum volume nor the minimum area of CAE is negative. The result of the division is shown in Figure 3.

Set the solver parameters, select the non-steady state calculation; set the solution model, select "k-epsilon (2eqn)"; customize the material properties according to Table 1.; set the regional conditions and boundary conditions, set the turbulence intensity and hydraulic diameter , making the export boundary conditions the same as the import; setting the dynamic grid to complete the pre-processing process.

Set the solution control parameters and solve the relaxation factor. The parameters of the relaxation factor are shown in Table 2. Convergence critical value is set, flow field initialization is completed, speed field animation is set, and iteration calculation is performed. The iterative calculation of the residual supervisor window is shown in Figure 4. As can be seen from the figure, the calculation is completed when iteratively calculates to 3548 steps.

Table 2. Relaxation Parameter Settings

Pressure	Density	Body Forces	Momentum	Turbulent Kinetic Enery
0.3	1	1	0.7	0.8

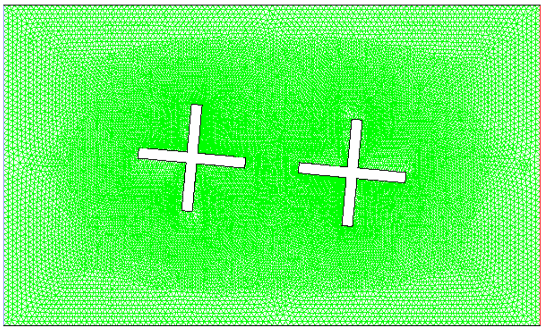


Figure 3. Schematic diagram of meshing results

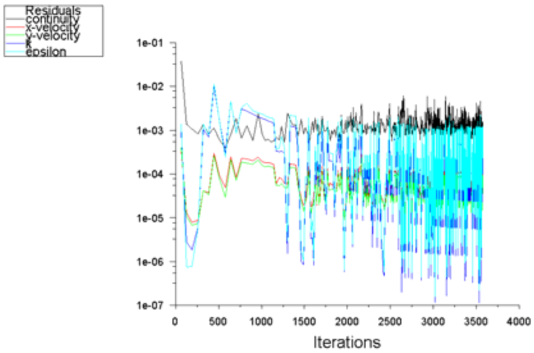
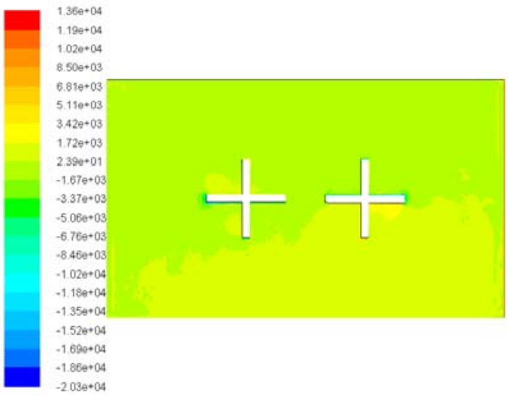
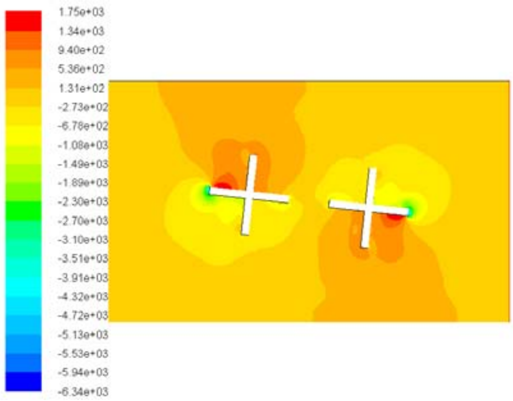


Figure 4. Residual Supervisor Window

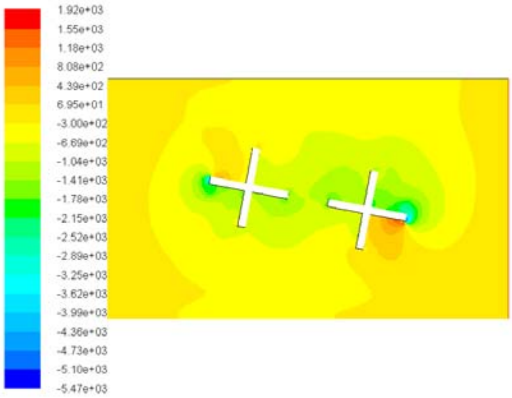
Figures 5 and 6 are the pressure cloud map and velocity cloud map of the simulation at 0.03s, 30s, 60s, and 90s. From the pressure cloud map and velocity cloud map, the two-rotor swirl flow field at each moment can clearly be seen clearly in the image. The pressure distribution and velocity distribution.



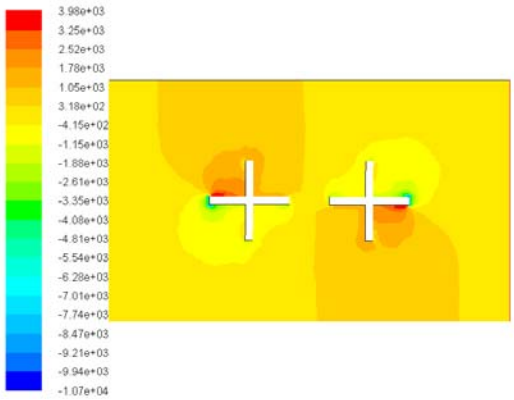
(a) Pressure cloud at time 0.03s



(b) Pressure cloud at time 30s

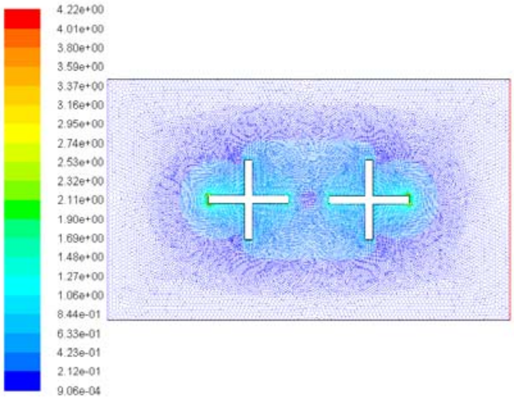


(c) Pressure cloud at 60 s

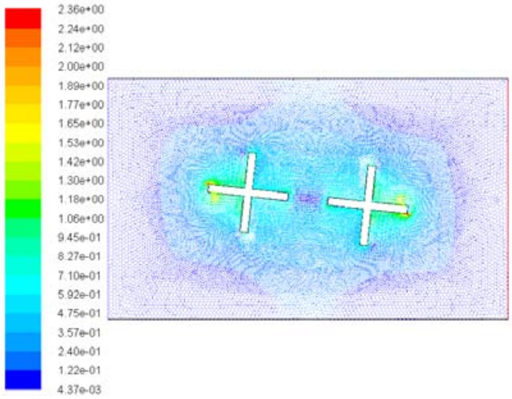


(d) Pressure cloud at 90 s

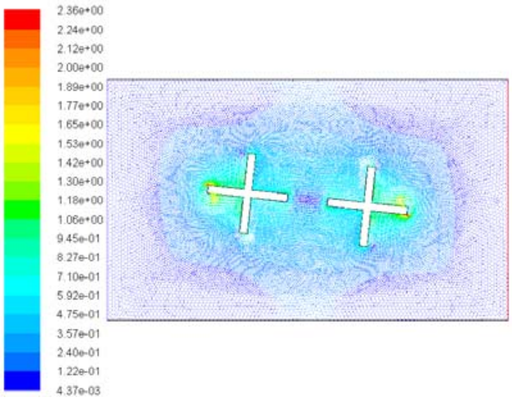
Figure 5. Pressure cloud at each time



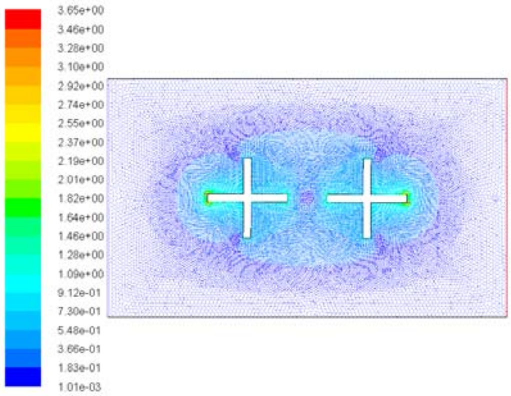
(a) Speed cloud at 0.03s



(b) Speed cloud at 30



(c) Speed cloud at 60 s



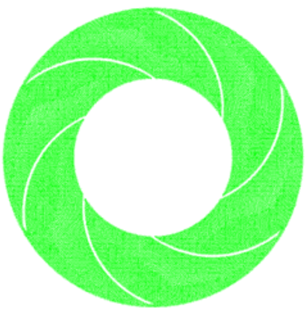
(d) Speed cloud at 90 s

Figure 6. Speed cloud at various times

2.3 Numerical Simulation of Internal Flow Field in Grouting Centrifugal Pump

Centrifugal pumps use the pre-processing software Gambit to mesh, initially establish boundary conditions and define internal surfaces. The two-dimensional simplified model of the centrifugal pump was numerically simulated using Fluent, and the two-dimensional simplified model of the centrifugal pump is shown in Figure 2.

Using the fluent rotating reference frame method, a slip coupling surface was created. The centrifugal pump diameter used for the analysis was 600 mm, the hub diameter was 300 mm, the number of blades was 6, and the rotation speed was 2900 r/min. Custom materials, in order to study the impact of different materials on the grouting stage, this simulation selected a new grouting material gypsum slurry and the traditional grouting material concrete slurry for simulation analysis and comparison. After completion of the preprocessing, setting of boundary conditions, setting of the solver, and setting of the solution method, iterative solution is performed. The residual error supervisor window in the iterative solution process is shown in Figure 8.



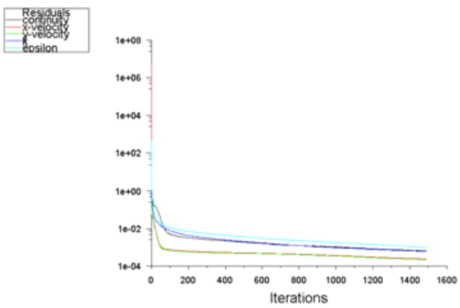
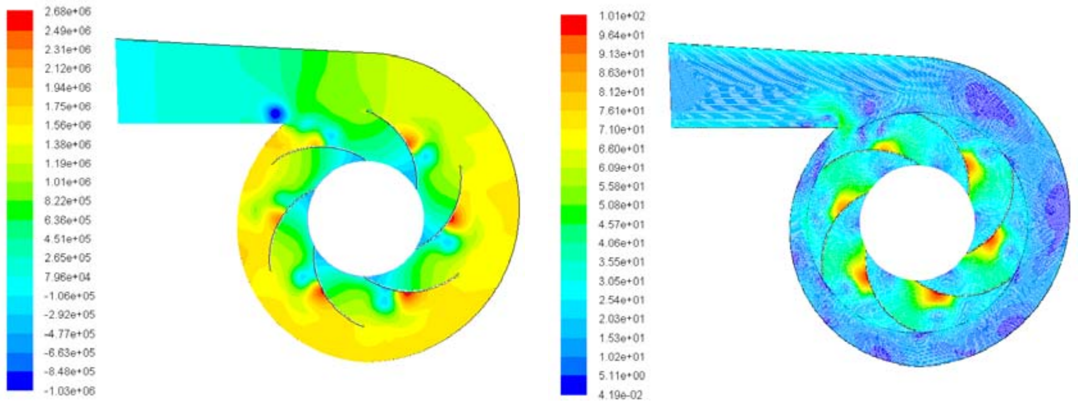
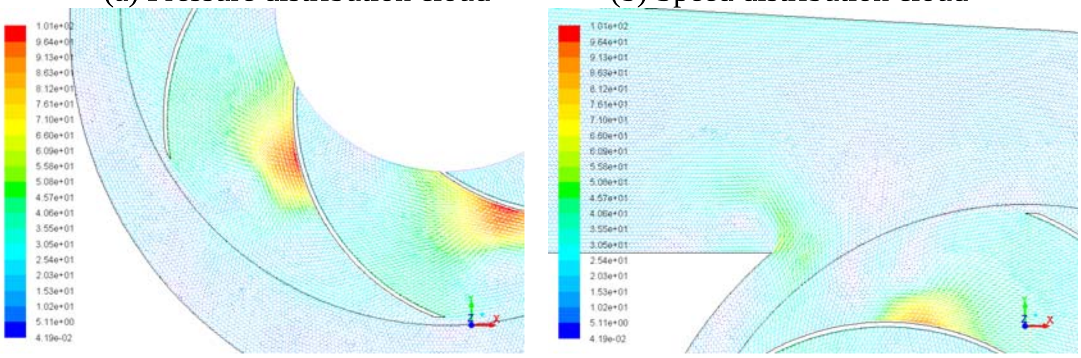


Figure 7. Meshing results of centrifugal pump 2D model Figure 8. Residual Supervisor window

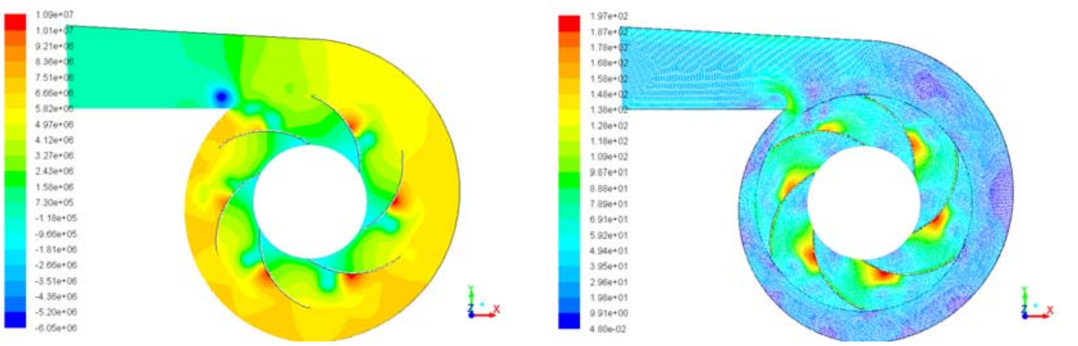


(a) Pressure distribution cloud (b) Speed distribution cloud

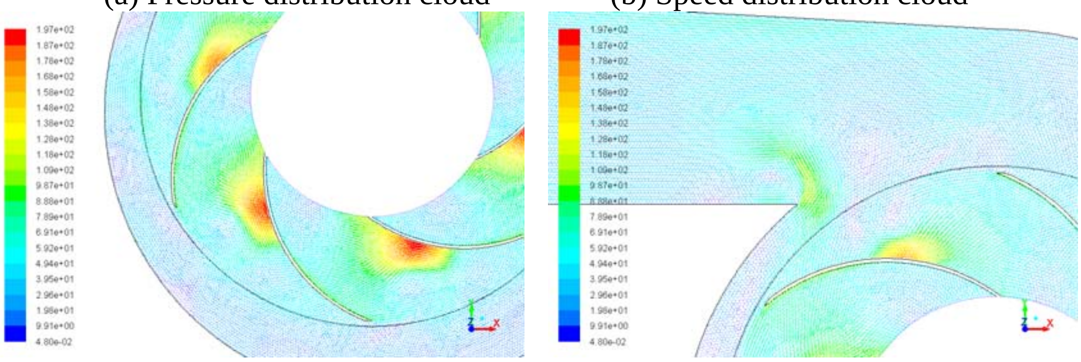


(c) Velocity vector cloud diagram of blade area (d) Speed vector cloud map of exit area

Figure 9. Distribution of Pressure and Velocity Fields for New Grouting Material



(a) Pressure distribution cloud (b) Speed distribution cloud



(c) Velocity vector cloud diagram of blade area (d) Speed vector cloud map of exit area

Figure 10. Distribution of pressure and velocity fields of traditional grouting materials

3. Flow Field Simulation Results Analysis

Through the use of the pre-processing software Gambit, the two-rotor rotary-row model and centrifugal pump two-dimensional model were meshed separately, and then the slip-coupled mesh model in Fluent was used to simulate the double impeller rotation in the mixing section and the centrifugal pump internal flow in the grouting section. At the same time, the influence of different materials on the pressure field and velocity field in the grouting section was also compared.

According to the simulation results, with the passage of time, the pressure around the impeller of the double impeller is most concentrated and shows an upward trend, and the velocity field exhibits similar properties. However, as the time increases to a certain extent, the velocity field tends to be stable. Therefore, the establishment of a reasonable time system can reduce the damage to the impeller. The difference in boring options during the grouting process has a great difference in the pressure caused by the centrifugal pump impeller. Therefore, under the condition of ensuring the safety of the wall, a new type of building material, desulfurized gypsum slurry mixture, can be selected to achieve the waste, energy saving and environmental protection, but also can greatly reduce the cost of raw materials.

At the same time, according to the flow field simulation analysis results, the distribution of velocity and pressure at different moments is analyzed. According to the reduction of the impact damage at the impeller outlet, the construction parameters can be optimized and the impeller outlet structure can be improved.

4. Conclusion

After analyzing the mixing and grouting links in the non-bearing wall cast-in-place process, the flow field inside the double impeller rotation and the centrifugal pump was simulated. The velocity field and pressure field in each time period were analyzed, and different wall materials were studied. With regard to the impact of each link, the advantages of the new type of pouring material are summarized and the optimized cast-in-place process parameters are obtained. For the future construction industry to optimize the construction program, improve the equipment structure to provide a certain amount of research ideas.

References

[1]. Li Dong. Towards Perceived Intelligent Buildings: Prospects for Future Development of Intelligent Building Industry [J]. Intelligent Building and Smart City, 2018(01):34-35.

[2]. Fulin. Analysis of Development Status and Future Strategy of China's Construction Industry [J]. China Economic and Trade Review, 2016(20):22-23.

[3]. Shen Zuyan, Luo Jinhui, Li Yuanqi. Taking the steel structure as a starting point to promote the coordinated development of the greening, industrialization and informatization of the construction industry [J]. Advances in Steel Structure Construction, 2016,18(02):1-6+25 .

[4]. Zhang Yao, Ren Wei, Liu Yulong, Zhang Zhaohui. Comprehensive Utilization of Sintered Desulfurized Gypsum [J]. Silicate Bulletin, 2015, 34 (12): 3563-3566+3570.

[5]. Qi Yanli. Research on performance and economic benefits of desulfurization gypsum masonry of green energy-saving wall materials[D]. Xi’an University of Architecture and Technology, 2015.

[6]. Liang Zhixi, Wang Haiyue, Ruan Xiangsong, Wang Jianpeng. Numerical simulation of flow field in pipeline based on Fluent software[J]. Agricultural Machinery of South China, 2017, 48(16):111.

[7]. Zhao Yue. Numerical Simulation of Pipeline Local Resistance Based on CFD[D]. Northeast Petroleum University, 2011.

[8]. GUO Jinsong, ZHANG Yongdong, FANG Fang, LONG Man, LONG Tiansong, LI Longjian. Numerical Simulation of Swirl Pool Separation Effect Based on Fluent Software[J]. Journal of Chongqing Jianzhu University, 2006(04): 79-82.

[9]. Park K M, Yoon H S, Min I K. CFD-DEM based numerical simulation of liquid-gas-particle mixture flow in dam break [J]. Communications in Nonlinear Science & Numerical Simulation, 2017.

[10]. Petitti M, Barresi A A, Marchisio D L. CFD modelling of condensers for freeze-drying processes[J]. Sadhana, 2013, 38(6):1219-1239.

[11]. Lei T, Zhu B S, Cao S L, et al. Cavitation flow simulation for a centrifugal pump at a low flow rate[J]. Chinese Science Bulletin, 2013, 58(8):949-952.

[12]. Liang Zhixi, Wang Haiyue, Ruan Xiangsong, Wang Jianpeng. Numerical simulation of flow field of pipeline based on Fluent software[J]. Agricultural Machinery of South China, 2017, 48(16):111.

[13]. Zhao Yue. Numerical Simulation of Pipeline Local Resistance Based on CFD[D]. Northeast Petroleum University, 2011.

[14]. GUO Jinsong, ZHANG Yongdong, FANG Fang, LONG Man, LONG Tiansong, LI Longjian. Numerical Simulation of Swirl Pool Separation Effect Based on Fluent Software[J]. Journal of Chongqing Jianzhu University, 2006(04):79-82.