Simulation Analysis of Flow Field Based on L-Shaped Pipe Injection

Xiaosong Huang[1, a], Pengwei Sun[1, b] and Wei Miao[1, c]

[1]School of Shandong University of Science and Technology, Shandong 266590, China.

[a]1129715585@qq.com, [b]1015791162@qq.com, [c]1351218987@qq.com

Abstract

With the advancement of society, the demand for some simple structures is increasing. At the same time, in terms of energy conservation, the efficiency of simple devices is also highlighted. The more important and important pipelines are closely related to our lives, and in production applications. There are a wide range of applications. So here I have a series of Fluent analysis simulations for the L-shaped pipeline. By using the K-turbulence model of fluent software, the flow field distribution simulation of the blow-shaped ventilation duct of the L-shaped pipeline from bottom to top was carried out, and the flow field state of the L-shaped pipeline under such working conditions was analyzed.

Keywords

L-shaped pipe, DPM, flow field distribution, Fluent.

1. Introduction

In this paper, the K-turbulence model is selected to carry out numerical simulation analysis on the flow field. The ICEM grid is divided into unstructured grids. Because the structure of the L-tube is relatively simple, the selection of the unstructured grid can not affect the premise of grid quality.[1] The calculation speed is greatly improved (including the solution of ICEM and Fluent).

2. Establishment of Physical Model

2.1. UG Modeling for L-Shaped Pipes.See Fig. 1.

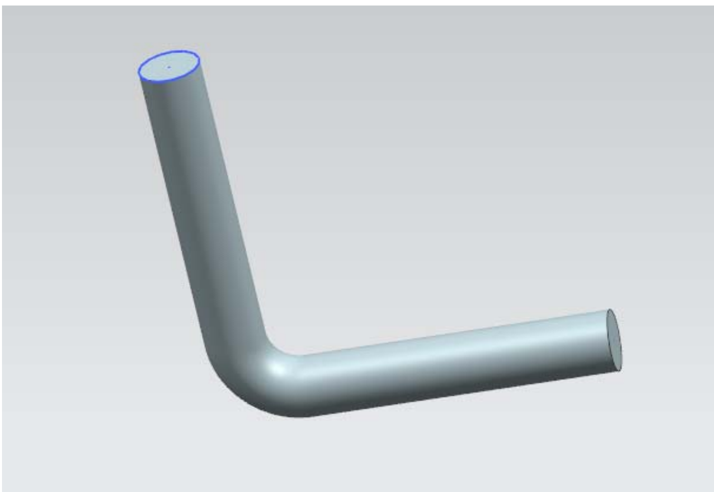

Figure 1. UG modeling for L-shaped pipes

2.2. Perform ICEM Meshing on the Model

The grid is divided into unstructured grids in ICEM, because in the L-tube, the structure of the device is relatively simple, so the unarchitected grid is used[2], because it can greatly speed up while maintaining the quality of the grid. The calculation time reduces the requirements on the computer and ensures the grid quality and calculation accuracy. see Fig. 2.

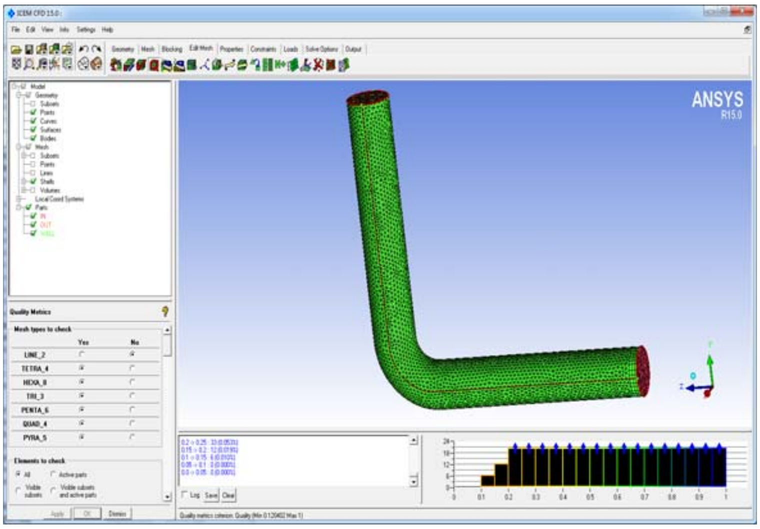

Figure 2. Perform ICEM meshing on the model

3. Perform Calculation and Analysis of Fluent on the Model

The data graph is obtained based on the Fluent calculation. The calculation equation for Fluent and the Reynolds number are calculated as

Calculate Reynolds number: $\mathrm{Re} = \dfrac{UD\rho}{\mu}$ Can be Re obtained as15000

Line chart, pressure, speed, temperature cloud map after Fluent.see Fig. 3.

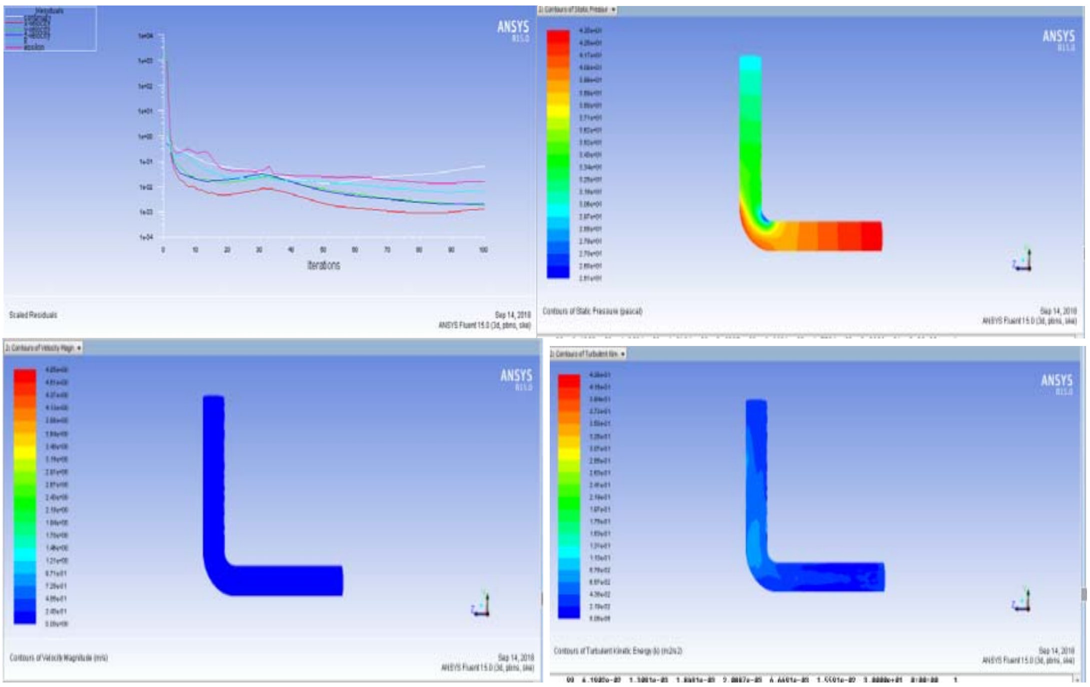

Figure 3. Perform calculation and analysis

3.1. For the Quality Inspection Results of Import and Export, It Is Shown That the Mass Conservation Law Is Followed, and the Mass Flow Map and Mass Distribution Map Are Obtained.In the L-Shaped Pipe, the Basic Governing Equation Is Used.

$$\int_{V(t)} \frac{\partial \rho(\mathbf{x},t)}{\partial t} dV + \int_{A(t)} \rho(\mathbf{x},t)\mathbf{u}(\mathbf{x},t)\cdot\mathbf{n}dA = \int_{V(t)} \left\{\frac{\partial \rho(\mathbf{x},t)}{\partial t} + \nabla\cdot\left(\rho(\mathbf{x},t)\mathbf{u}(\mathbf{x},t)\right)\right\} dV = 0 \quad \iiint_V \nabla\cdot\mathbf{Q}dV = \iint_A \mathbf{n}\cdot\mathbf{Q}dA$$

3.2. Track the Droplet Particles to Get the Corresponding Droplet Trajectory Distribution.from the Cloud Image of the Tracking Particle, You Can Find the Distribution Trajectory of the Particle in the Pipeline.See Fig. 4.

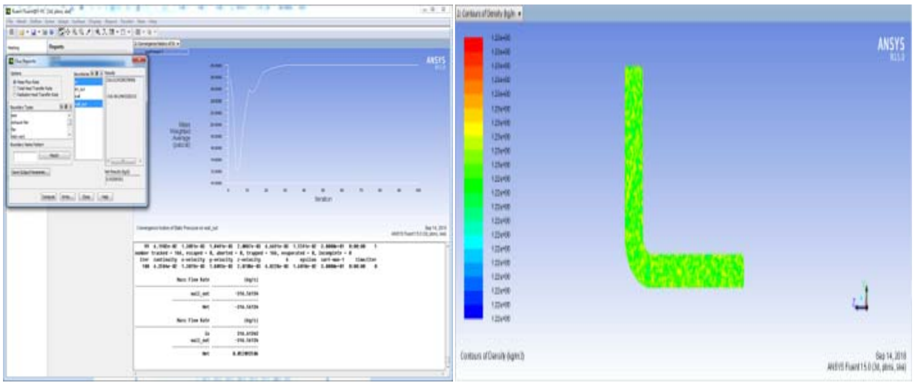

Figure 4. Track the droplet particles

3.3. Monitoring of Discrete Items of Convective Items

Two-phase flow: A gas or liquid flow containing a large amount of solid or liquid particles is usually referred to as a two-phase flow; a group of particles having a plurality of sizes is one "phase", and a gas or liquid is another "phase", thereby There are gas-liquid[3], gas-solid, liquid-solid and other two-phase flow. The eddy current phenomenon occurs at the bottom of the L-shaped tube because when the particles enter from the inlet and quickly collide with the bending tube, a rebound occurs, and a part of the eddy current phenomenon is formed.see Fig.5.

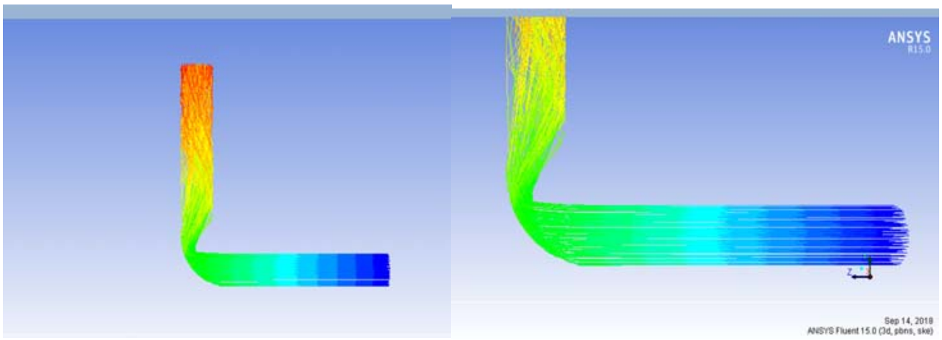

Figure 5. Two-phase flow

According to the particle tracking of DPM, the initial pressure is faster after the particles enter the pipeline, but when the impact bends, a series of energy losses are generated, resulting in a decrease in pressure and speed. Part of the energy loss of the particle during the collision [4].

4. Conclusion

In this paper, the K-turbulence model is selected to carry out numerical simulation analysis on the flow field. The ICEM grid is divided into unstructured grids. Because the structure of the L-tube is relatively simple, the selection of the unstructured grid cannot affect the premise of grid quality. [5]The calculation speed is greatly improved (including the solution of ICEM and Fluent). The flow field of the L-tube is numerically predicted and the inlet is numerically predicted. The DPM discrete phase tracking is added to the fluent setting to further understand the flow state of the particles in the L-tube. Simulate the state of the reaction. The calculation results are expressed in the form of a cloud image, which is more clear and easy to understand. By observing the pressure and velocity of the L-tube, it can be found that the pressure at the inlet is relatively large, and with the subsequent progress, the pressure it is progressively decreasing. The calculations show that the particles rebounding when hitting the elbow is large and affects the velocity of the particles, that is to say, the bending process deals with a large part of the energy loss of the particles during the collision. There may be some degree of instability for the analysis in the pipeline. To obtain a convergent solution, the relaxation factor must be reduced, which is followed by more calculation time and number of iterations. After considering various factors comprehensively: the calculation accuracy and calculation time obtained are not reasonable, and it is necessary to calculate the problem using a higher precision and time-sensitive format [6].

The change of the grid number will also affect the calculation result. The more mesh division, the closer the calculation result is to the reference solution [7], and the denser the mesh division, the worse the timeliness and the more iterations, so the grid is divided. [8] When we need comprehensive accuracy and timeliness, it is also very important to determine a reasonable number of grids. The quality of the grid determines the surprise of the calculation to a certain extent.

References

[1] Wang Ruihe, Li Luopeng, Zhou Weidong et al. Experimental and engineering calculation model of abrasive jet rotary cutting casing[J]. Journal of China University of Petroleum: Natural Science Edition, 2010, 34(2): 56-61.

[2] Huang Zhongwei, Li Gensheng, Shi Huaizhong, et al. Experimental study on abrasive jet jet casing and limestone under confining pressure[J]. Journal of China University of Petroleum: Natural Science Edition, 2014, 38(6): 85-89.

[3] Hreha P, Radvanska A, Knapcikova L, et al. Roughness Parameters Calculation By Means Of On-Line Vibration Monitoring Emerging From AWJ Interaction With Material [J]. Metrology and Measurement Systems, 2015, 22(2): 315 -326.

[4] Hu Dong, Xia Zhihua, Tang Chuanlin. Research on Post-mixed Abrasive Water Jet Cutting System[J]. Machine Tool & Hydraulics, 2013, 41(19): 33-35.

[5] Zhu Jianjian. Design of abrasive jet windowing device for oil and gas pipelines [D]. China University of Petroleum: China University of Petroleum, 2011.

[6] Guo Zonghuan. Experimental study on micro-abrasive water jet cutting polysilicon [D]. Xihua University: Xihua University, 2014.

[7] Fan Jirui. Study on continuous jet cutting characteristics of deep sea high pressure abrasive [D]. Dalian Maritime University, 2013.

[8] Zhou Dapeng, Duan Xiong, Jiang Shan. Experimental study on the relationship between cutting depth and velocity of post-mixed abrasive jet under different cutting pressures[J].Coal Mine Machinery,2013,5:042.