ISSN: 2664-9640

Multi-channel pipeline flow field analysis

Shuhao Pan[1], Kelun Tang[2], Huaqian Zhang[1] and Chi Zhang[1]

[1]School of Mechanica Engineering University of Science and Engineering. Yibin 64400, China;

[2]School of Civil Engineering, Architecture and Environment Science, Xihua University, Chengdu 611730, China

Abstract

Fluent is a relatively popular CFD software in the world, which can be used to realize finite element calculation of fluid, heat transfer and chemical reaction problems. This paper takes the flow field analysis of multi-channel pipelines as an example to introduce the application of fluent. A multi-channel pipeline model is established, assuming that the pipeline is a mixed flow of cold and hot flows, and the calculation results are post-processed by the post-processing software of fluent to analyze the distribution of flow velocity, temperature, and pressure in the flow field.

Keywords

Fluent; Multi-channel pipeline; Flow field analysis; Finite element.

1. Introduction

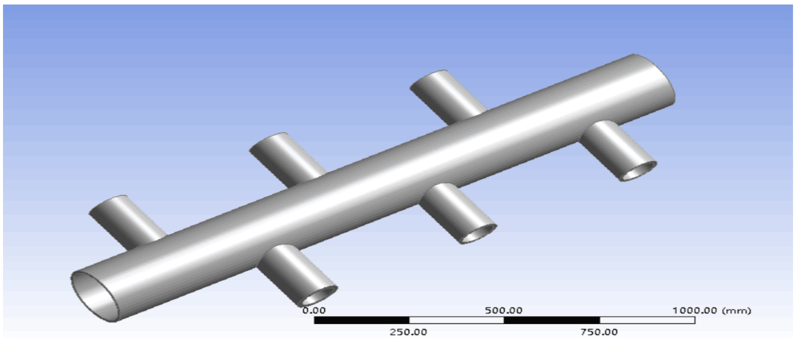
The development of multi-channel pipelines is relatively mature at present, and the scope of application is relatively wide. The use of multi-channel pipelines helps to improve the efficiency of pipeline transportation. Combined with finite element knowledge for analysis. By analyzing the flow of heat flow and cold flow in the pipeline, the influence of the fluid region on the pipeline can be determined. Through the analysis of the flow field distribution, the flow of the fluid in the multi-channel pipeline and the influencing factors, the flow rate of the thick pipe and the flow rate of the thin pipe can be known. The relationship between the temperature in the pipe and the influence of the thick pipe and thin pipe fluid, and the relationship between the pressure distribution in the pipe and the fluid flow[1-2].

Using fluent to analyze the flow field of the model in ANSYS is the process of numerically solving the established numerical model, and the finite element is the most basic and extensive theoretical basis in the field of engineering simulation experiments [3-4]. The law of any fluid motion is based on the law of conservation of mass, the law of conservation of momentum and the law of conservation of energy [5-6]. These basic laws can be described by mathematical equations, such as Euler's equation, N-S equation. Using numerical calculation method to solve these mathematical equations that control fluid flow through computer and then study the motion law of fluid, this is the application of finite element knowledge. In terms of actual fluid dynamics, the use of general CAE software on the basis of finite element to complete the analysis of some fluid mechanics problems in engineering has a very broad application prospect. In recent years, with the development of computer technology and related technologies, finite element technology has been greatly improved, and significant progress has been made in the field of engineering, especially in the fields of magnetic field, electric field, stress field, flow field, etc. effect.

The multi-channel pipeline analyzed in this paper is widely used in actual production and life, and can be used for the mixing of liquids of different properties, the mixing of liquids of different temperatures, etc. This pipeline model is relatively simple, and it is easy to carry out systematic flow field analysis of the pipeline. It is more helpful to be familiar with fluent software and the application of finite element theory knowledge.

2. Multi-Channel Pipeline Model

The fluid dynamics exercise model used in this exercise is a multi-channel pipe with a relatively simple structure, which consists of a main thick pipe and six branched thin pipes. The structural model is shown in Fig 1. The internal flow medium of the multi-channel pipeline model is designed to be water, and it is assumed that the pipeline is a mixed flow of cold and hot flows. The total length of the thick tube is 1800mm, the length of the thin tube is 200mm, the inner diameter of the thick tube is 100mm, the outer diameter is 110mm, the inner diameter of the thin tube is 50mm, and the outer diameter is 60mm. The design pipeline heat flow inlet velocity is 10m/s, the temperature is 500K, the cold flow inlet velocity is 20m/s, the temperature is 300K, and the outlet is a free outlet.



Fig, 1 3D model of pipeline

2.1. Model meshing

Use the preprocessing software to mesh the fluid area, and the fluid area model is shown in Figure 2. When dividing the mesh, it should be noted that in the fluid analysis, other models except the fluid model are not involved in the calculation, so it needs to be suppressed when doing the fluid analysis [7-8]. When doing fluid analysis at the same time, it is generally necessary to set the expansion layer to mesh the geometry. In order to simplify the meshing step, unstructured meshing is adopted, which can more effectively adapt to complex solution domains with irregular shapes and curved boundaries. In order to ensure the computational efficiency, this model is divided by tetrahedral elements. Since the model is a circular pipeline, the grid boundary layer is set to 2 layers, resulting in a total of 218,803 nodes and 1,036,754 grid elements. The mesh model and mesh quality after the division is completed are shown in Fig2 and 3

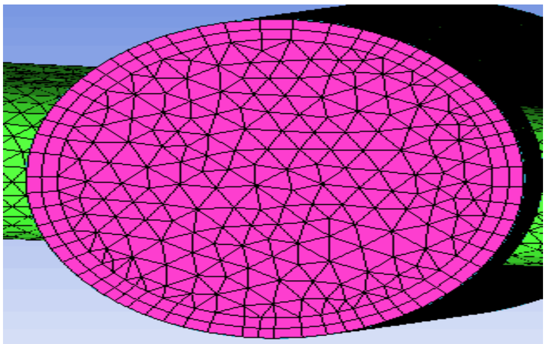


Fig. 2 Fluid Domain Model

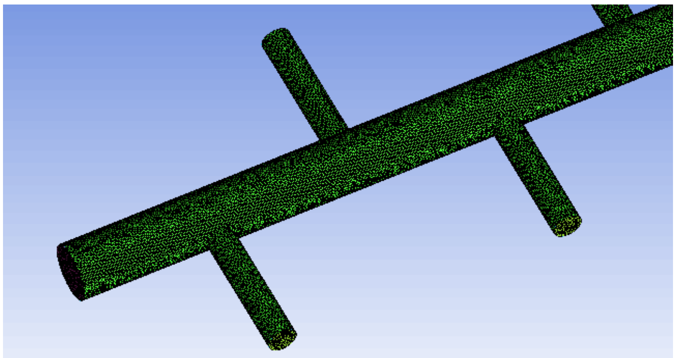


Fig. 3 Model Meshing

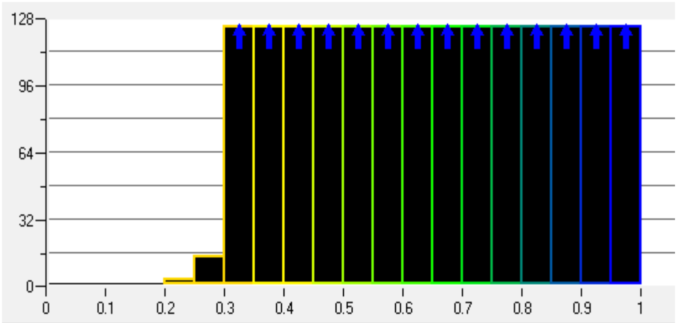


Fig.4 Grid Quality Map

2.2. Flow state analysis

In fluid dynamics calculations, it is first necessary to determine whether the flow state of the simulated flow system is transient or steady, and then define whether the class of the flowing fluid is viscous or non-viscous. Inviscid fluid flows are generally compressible, and the physical properties of flow compression can be addressed by the plate method. The viscous fluid flow is divided into two states: laminar flow and turbulent flow. The flow medium of this pipe model is water, which is incompressible and therefore a viscous fluid. In the process of analysis and calculation, the analysis and calculation are different for different flow states, so it is necessary to judge whether the flow is laminar or turbulent. According to the Reynolds number, if the Reynolds number is less than 2300, it is laminar, and if the Reynolds number is greater than 2300, it is turbulent[9].

In this calculation model, the inner diameter of the thick tube is D_1=100mm, the inner diameter of the thin tube is D_2=100mm, the inlet velocity of the thick tube is U_1=20m/s, the inlet velocity of the thin tube is U_2=10m/s, and the Reynolds number is calculated as below equation.

$$R_e = \frac{\rho U L}{\mu} \tag{1}$$

where: R_e is the Reynolds number; ρ is the density, the density of water is$\rho=1Kg/m^3$; U is the velocity of the fluid in the pipe; L is the effective length, in the pipe model, L=D, D is the inner diameter of the pipe; μ is the kinematic viscosity of water, the kinematic viscosity of water is $\mu=1.01\times10^{-3}Pa\cdot s$;

The thick tube Reynolds number is calculated as follow,

$$R_{e1} = \frac{\rho U L}{\mu} = \frac{1\times10\times0.1}{1.01\times10^{-3}} = 990.1 \tag{2}$$

The thin tube Reynolds number is calculated as follow,

$$R_{e2} = \frac{\rho U L}{\mu} = \frac{1\times20\times0.05}{1.01\times10^{-3}} = 990.1 \tag{3}$$

It can be seen from the calculation that if the Reynolds number is less than 2300, the flow state of the fluid in the pipeline is judged to be laminar flow.

2.3. Computational Numerical Model

In this paper, the distribution and variation of the internal flow field of the multi-channel pipeline model are studied. The fluid calculation must satisfy the mass conservation equation, that is, the continuity equation. The expression of the continuity equation is:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{4}$$

where: ρ is the density, kg/m^3;**t** is the time period;µ is the velocity vector,s/m;u v w is the component of the velocity vector in the x,y and z directions.

2.4. Boundary Condition Analysis

The complex characteristics of fluid flow behavior are included in the boundary conditions that describe the flow problem. Appropriate boundary conditions are the physical characteristics of simulating real fluid flow, and are also a key step in the process of fluid analysis. This model studies the internal flow field and pressure distribution of hot and cold fluids flowing in multi-channel pipes. The boundary conditions need to set the inlet and outlet of the fluid, the velocity of the inlet and outlet, and the temperature and pressure of the fluid at the outlet and other attributes [10]. The specific physical parameters and related parameters are set as follows.

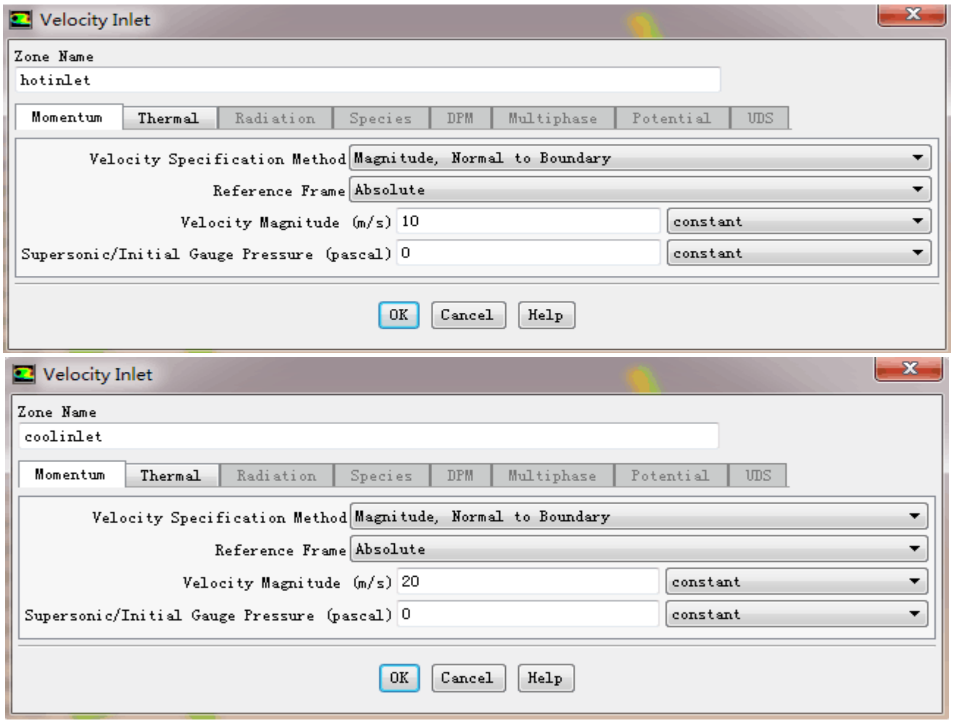
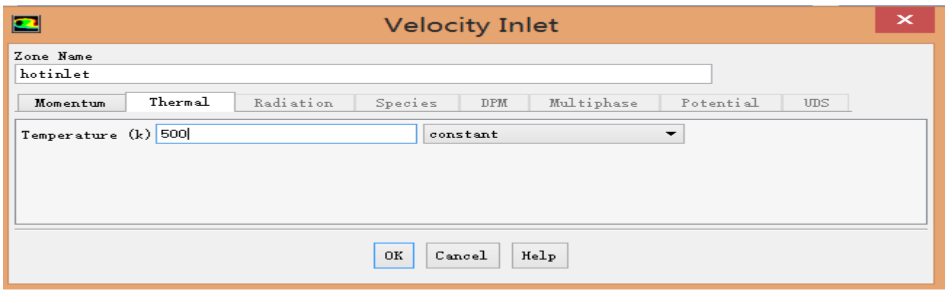


Fig 5 Inlet Velocity Parameters



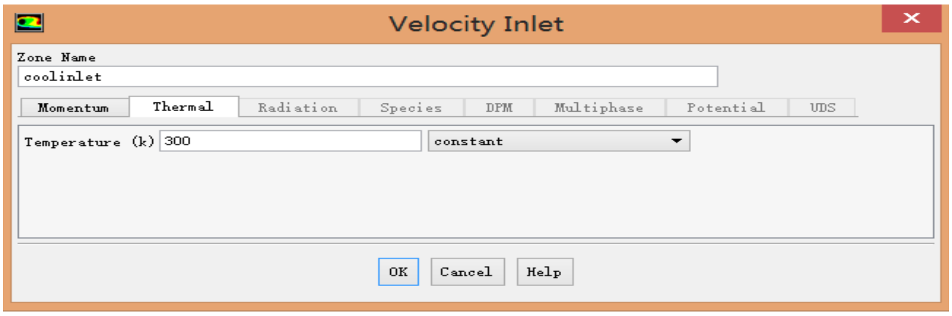


Fig 6 Inlet temperature parameters

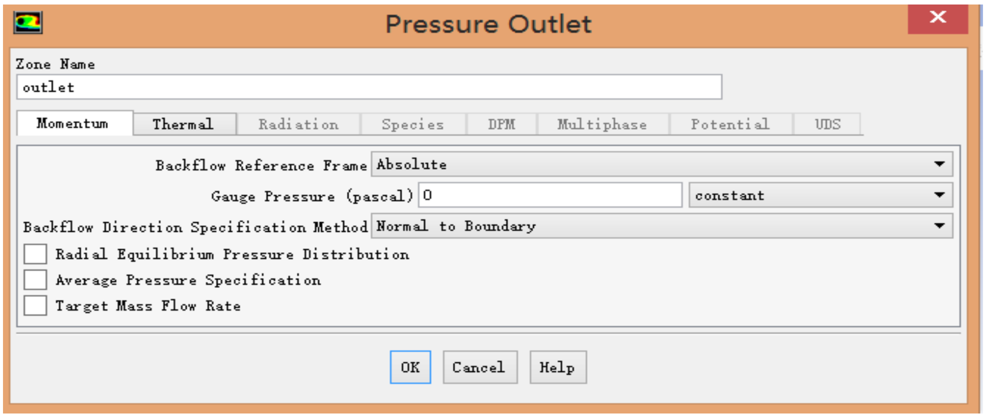


Fig 7 Outlet pressure setting

3. Result post-processing and flow field analysis

The post-processing of the results is an important manifestation of the visualization in the calculation result report of the flow field analysis. The calculation results are post-processed by fluent's own post-processing software to analyze the distribution of flow velocity, temperature and pressure in the flow field. Influencing factors and changing laws. Post-processing can generate vivid graphical results, and fluid changes can be analyzed by different colors.

3.1. Velocity Field Analysis

Figure 8 is the cloud map of flow velocity distribution, and figure 9,10,11 are the velocity distribution in each direction of XYZ. It can be seen from the figure that there is a certain regularity in the velocity of the fluid in the pipeline, and the velocities of the thick tube and the thin tube influence each other. The inlet velocity is small and stable, and the flow velocity increases locally in the pipeline due to the fluid interaction between the thick and thin tubes at the intersection. The closer the position of the pipeline intersection, the greater the flow velocity. It can be clearly seen that the change of the flow velocity of the thick pipe is greatly affected by the thin pipes on both sides, indicating that the flow velocity is affected by other factors. The flow velocity at the inlet of the thin tube is relatively stable. Before it is not mixed with the fluid in the thick tube, the flow velocity of the thin tube is not affected by the fluid in the thick tube. the flow rate in the thick pipe increases, so the closer to the end of the thick pipe, the greater the flow velocity at the outlet of the thin pipe.

As shown in the figure 9 below, from the flow velocity cloud diagram in the X direction, the flow velocity at each position in the pipeline is almost the same, indicating that the flow field in the entire pipeline in the X direction is relatively stable and is not affected by the interaction of other factors.

The velocity of the thin tube fluid in the Y direction is greater than that of the thick tube fluid, and the flow velocity of the thick tube fluid is less affected.

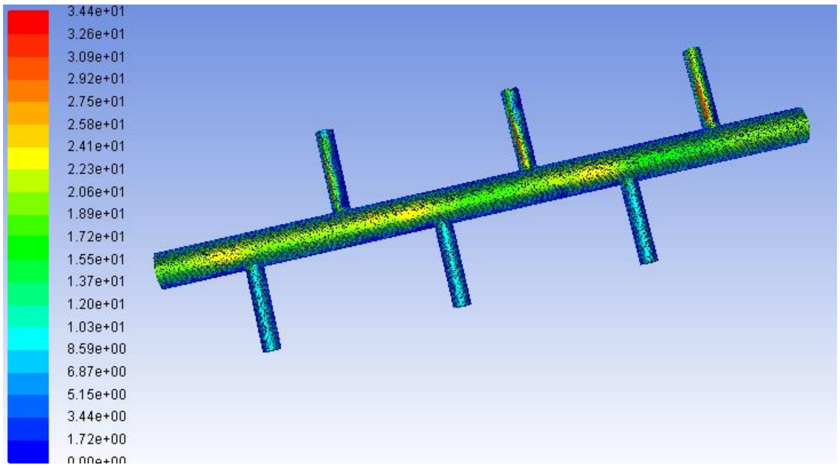


Fig 8 Velocity cloud map

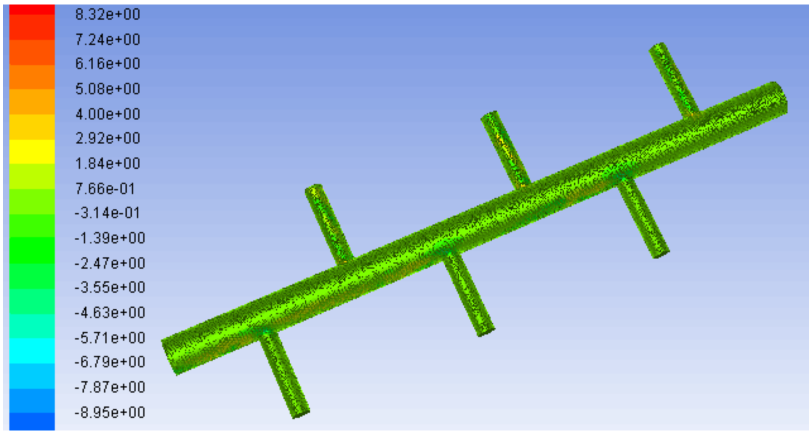


Fig 9 Flow velocity cloud map in X direction

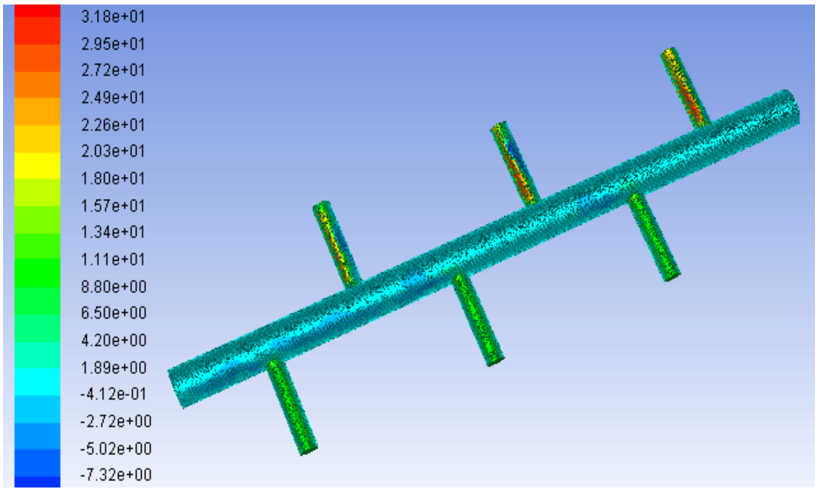


Fig 10 Cloud map of flow velocity in Y direction

It can be seen from the Z-direction velocity cloud diagram in the figure below that the velocity of the thick pipe in the Z direction is affected by the flow velocity of the thin pipe. When only the fluid in the thick pipe flows, the velocity is larger. When the fluid in the thin pipe is mixed, the thick pipe fluid is affected by the impact, and the velocity somewhat reduced. The flow velocity at the inlet of the thin tube and the flow velocity at the outlet of the thin tube show a similar flow law, and the flow velocity in the thin tube is generally greater than that of the thick tube.

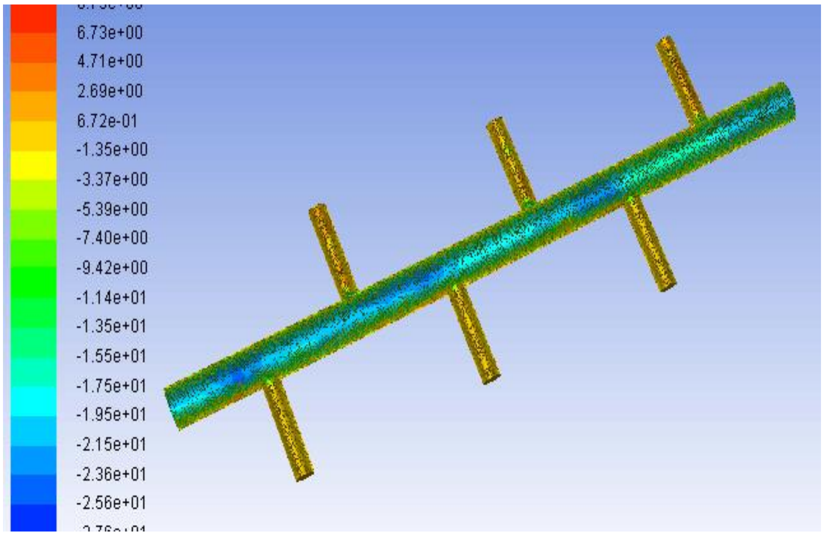


Fig 11 Flow velocity cloud map in Z direction

3.2. temperature field analysis

Figure 12 is the temperature vector cloud map, which can reflect the temperature distribution in the pipeline flow process. In the heat flow inlet pipe, the temperature of the fluid is relatively high due to the small influence of the cold flow. As the heat flow flows into the thick pipe, the flow rate of the cold flow is greater than that of the heat flow, so the temperature of the heat flow changes rapidly. The temperature of the fluid decreases gradually due to the continuous increase of the cold flow at the cold flow inlet of the thick tube and the heat loss of the heat flow during the flow process. In the inlet of the thin tube, the fluid is not affected by the temperature of other factors. In the thick tube, the cold flow and the hot flow are in a mixed state and affect each other.

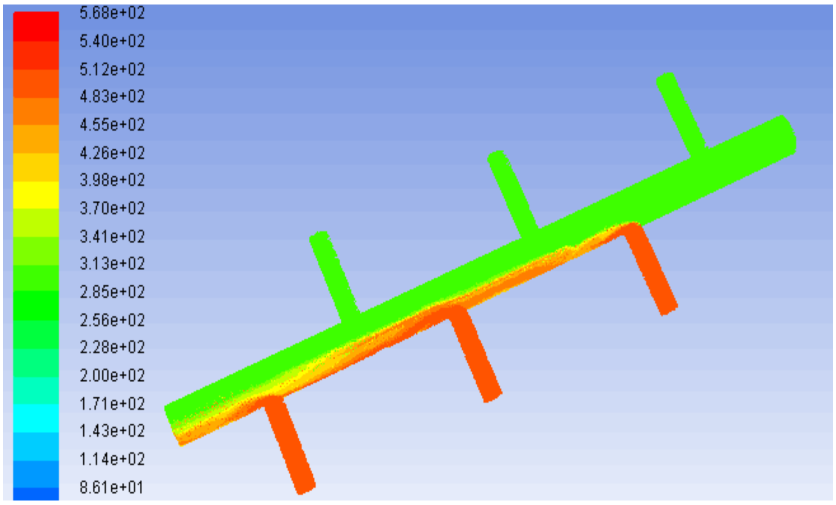
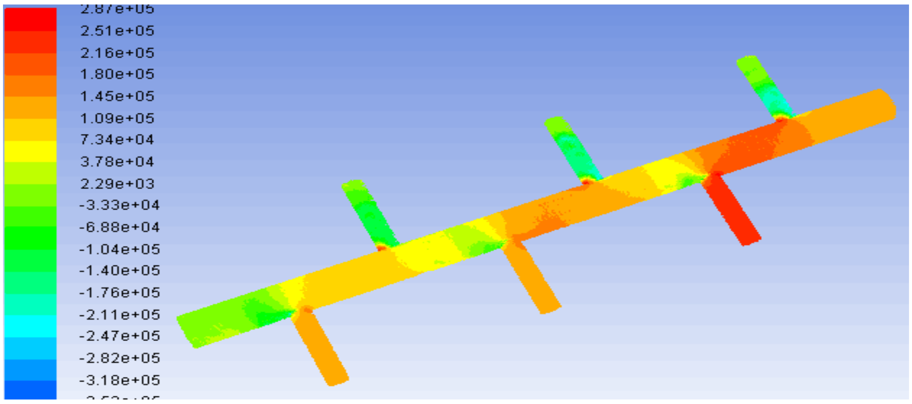


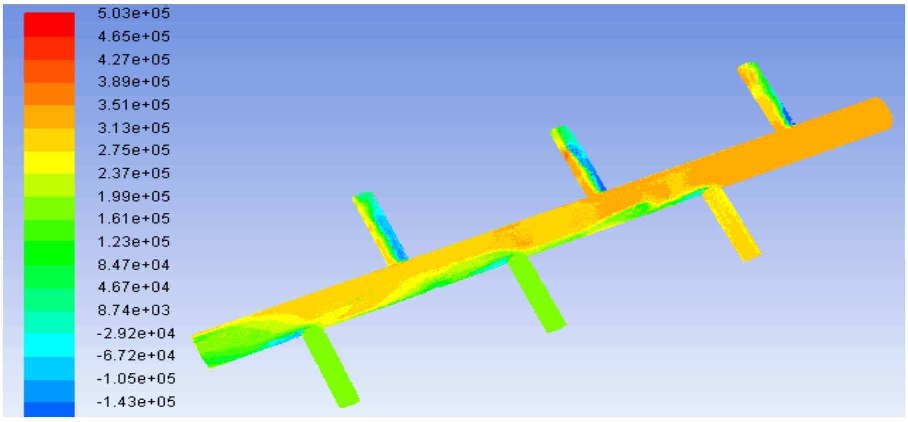
Fig 12 Temperature vector cloud map

3.3. Pressure Field Analysis

Figure 13 shows the internal pressure distribution cloud map of the fluid pipeline, (a) is the static pressure cloud map, (b) the overall pressure distribution cloud map. It can be seen from the figure that the static pressure distribution of the thick pipe is greatly affected by the fluid flow, and the inlet of the thin pipe close to the outlet end is subject to greater pressure. Secondly, the inner wall of the thick pipe on the side of the thin pipe inlet is affected by the impact of the thin pipe inlet, and is affected by the impact of the thin pipe inlet. relatively high pressure. The static pressure at the outlet of the three thin tubes is less affected by the inlet flow rate and flow velocity of the pipe. From the cloud map of the total pressure distribution, the inside of the pipeline is generally mainly affected by the flow velocity and flow of the fluid. When the flow increases, the pressure inside the pipeline increases. The pressure distribution of the three thin tube outlets is greatly influenced by other factors. The closer to the end of the pipe, the greater the fluid pressure it receives due to the increase of fluid flow in the pipe.



(a) Static pressure distribution cloud map



(b)Total pressure distribution cloud map

Fig 13 Pressure distribution cloud map

4. Summary

In this paper, a simple flow field analysis of the multi-channel pipeline model is carried out, and the distribution of the velocity, temperature and pressure of the flow field inside the pipeline is mainly analyzed. Combined with the structure of the pipeline model, the fluid region model is established, and the mesh is divided by the preprocessing software. The analyzed model is solved and calculated by fluent software, the numerical simulation of the flow field distribution in the pipeline is realized, and the distribution of the velocity field, temperature field and pressure field in the pipeline fluid region is obtained.

Through the analysis of the flow field distribution, it can be known that the fluid flow in the multi-channel pipeline is influenced by each other, and the flow velocity of the thick pipe is greatly affected by the flow velocity of the thin pipe. The size of the temperature in the pipe is mainly affected by the thick pipe fluid, and the temperature distribution is relatively stable. The pressure distribution is affected by the fluid flow. As the fluid inlet increases, the flow increases and the fluid pressure on the pipeline increases. The processing of the analysis results can promote the design of related types of pipeline structures.

References

[1] Jiang Xiaofang, Cao Xijing, Si Zhenpeng. Application of FLUENT Technology in Industrial Pipeline Design [J]. Chemical Equipment and Pipeline, 2009, 46(5):3.

ISSN: 2664-9640

[2] Zhang Yu, Luan Jiangfeng, Zhang Siliang. Analysis of internal flow field of pressure pipeline based on FLUENT [J]. Contemporary Chemical Industry, 2014, 43(6):3.
[3] Wu Guangzhong, Song Tingting, Zhang Yi. FLUENT Basic Introduction and Case Mastery [M]. Electronic Industry Press, 2012.
[4] Ding Xinshuo, Ling Guilong. ANSYS Workbench 14.5 Finite Element Analysis Case Details [M]. Tsinghua University Press, 2014.
[5] JiyuanTu, GuanHengYeoh, ChaoqunLiu, et al. Computational Fluid Dynamics: Learning from Practice [M]. Northeastern University Press, 2014.
[6] Yuqing Zhang, Jun Liu. CFD Analysis of Free Falling Ball Osmometer in Clay. Springer International Publications, 2018.
[7] Yao Anlin, Hu Yiping, Zeng Xiangguo, et al. Finite element numerical analysis of crack interaction in high pressure gas pipelines [J]. Pressure Vessel, 2011, 28(3):5.
[8] Long Anhou, Zhang Fan, Han Shuai. Numerical simulation of temperature drop of submarine oil pipeline based on Fluent [J]. Science Technology and Engineering, 2011, 11(34):4.
[9] Xiong Lifang, Lin Yuan, Li Shiwu. - Turbulence model and its application in FLUENT software [J]. Industrial Heating, 2007, 36(4):3.
[10] Wang Changbin, Zhao Yue, Jia Xuesong, et al. Numerical simulation of temperature field of hot oil pipeline [J]. Pipeline Technology and Equipment, 2011(3):4.