ISSN: 2664-9640

Research on Flow Distribution of Porous Tube Based on Fluent Fluid Simulation

Junfeng Xue[1], Xiaosheng Song[2] and Jiuwei Wang[3]

[1] North China University of Science and Technology, Tangshan 063021, China;

[2]North China University of Science and Technology, Tangshan 063021, China.

[3]Hebei Zhen'an Seismic Isolation Technology Co., Ltd. Tangshan 063009, China.

Abstract

In order to study the inflow and outflow process of the pressurized fluid and the flow distribution of each hole in the two cavities connected by the conduit, the fluid region model of the cylinder is established, and Fluent fluid simulation analysis is used. It monitors the mass flow of each hole at different intervals, and calculates the proportion of the flow of each hole in the total inflow and outflow flow. The results show that the proportion of flow into each hole has a functional relationship with the position of the hole, and the calculation error of the curve obtained by fitting the inflow flow does not exceed 8.09%. The flow out of each hole is relatively uniform, and the calculation error according to the average distribution does not exceed 5.97%.

Keywords

Fluent fluid simulation; porous outflow; flow law; curve fitting.

1. Introduction

In order to realize the circulation of liquid between two adjacent closed cavities of a mechanical device, a conduit with a hole in the side wall is designed. Two adjacent cavities can be connected through the conduit, and the liquid flows between the cavities through the conduit [1]. Both ends of the duct are closed, and a plurality of holes are opened on the side wall of the duct with the center of the length of the duct as the axis of symmetry, and the flow rate can be controlled by controlling the number of openings. The structure of the duct is shown in Figure 1.

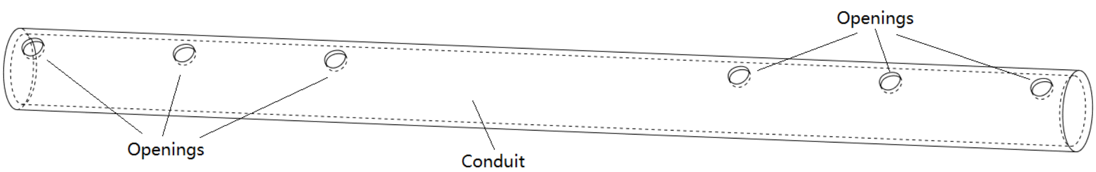

Fig. 1 Schematic diagram of catheter structure

The two cavities on the outside of the catheter are closed relative to the outside world. When the device enters the working state, the cavities on both sides will shrink and expand to a certain extent. During this process, the liquid in the shrinking cavity will pass through the side wall of the catheter due to the squeezing effect. The opening on the upper part enters the interior of the catheter, and flows through the cavity of the catheter into the dilation cavity from the opening on the other side. Since the cavities on both sides will repeatedly contract and expand, the liquid on both sides of the catheter will also flow back and forth in the catheter through the through holes on the catheter. A three-dimensional cross-sectional schematic diagram of the catheter and the closed cavity device is shown in Figure 2.

ISSN: 2664-9640

In order to meet the requirements of various working states and working efficiency of the mechanical device, it is necessary to design pipe fittings with various opening spacings, and study the fluid flux distribution relationship of each hole under different opening spacings. However, since each opening is only a few millimeters in size, it is difficult for sensors such as flowmeters to be connected, and the sensor will also affect the flow state of the fluid in actual work. What is more troublesome is that the cavities on both sides of the pipe fitting are in a closed space, and sensors are added. It will definitely affect the sealing of the fluid space, so it is difficult to measure the real flow data. Based on the above reasons, we will choose the Fluent fluid simulation software to simulate the flow of the liquid inside the conduit and the two sides of the cavity during operation. At the same time, the software can easily monitor the flow data of each opening [2].

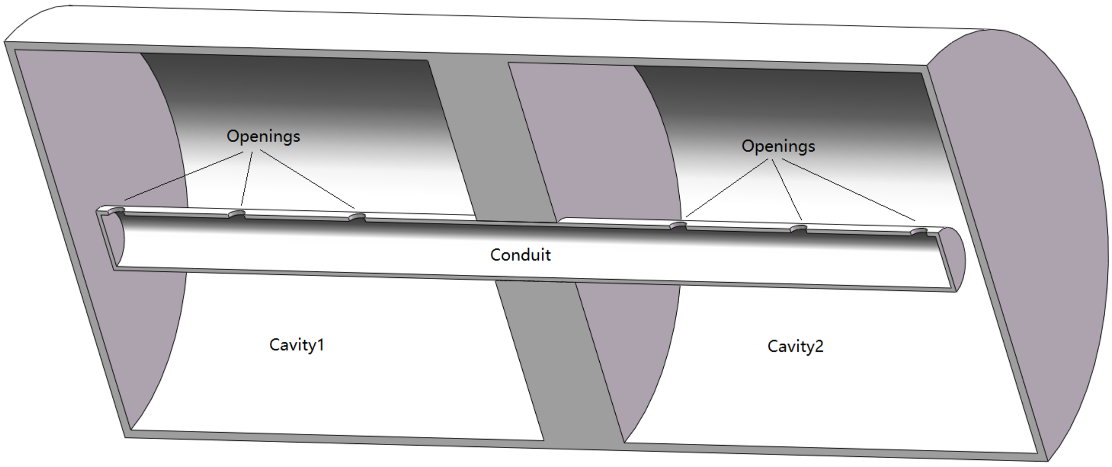

Fig. 2 Three-dimensional cross-sectional schematic diagram of catheter and closed cavity device

2. Theoretical model derivation

Assuming that the fluid is an incompressible fluid, according to the fluid continuity equation [3], the relationship between the total outflow or inflow flow in the conduit and the flow flux of each opening is as follows:

$$Q_a = Q_1 + Q_2 + \cdots + Q_n \tag{1}$$

In the above expression, Q_a represents the total inflow or total outflow flow in the fitting, $Q_1, Q_2, \cdots, Q_n$ represents the flow rate of each inflow or outflow hole, n represents the number of inflow or outflow holes.

Substitute $Q = V \cdot \mathrm{A}$ into formula (1) to get

$$V_0 \cdot \mathrm{A}_0 = V_1 \cdot \mathrm{A}_1 + V_2 \cdot \mathrm{A}_2 + \cdots + V_n \cdot \mathrm{A}_n \tag{2}$$

In the above expression, V_0 represents the velocity at the inflow boundary of the external fluid domain, A_0 represents the area of the inflow boundary of the external fluid domain, $V_1, V_2 \cdots V_n$ represents the flow velocity at each inflow or outflow hole, $\mathrm{A}_1, \mathrm{A}_2 \cdots \mathrm{A}_n$ represents the area of each inflow or outflow hole.

Since each opening has the same area, that is

$$\mathrm{A}_1 = \mathrm{A}_2 = \cdots = \mathrm{A}_n \tag{3}$$

Therefore, (2) can be simplified as

$$V_0 \cdot \mathrm{A}_0 = \mathrm{A}_k (V_1 + V_2 + \cdots + V_n) \tag{4}$$

which is

$$V_0 \cdot \frac{A_0}{A_k} = V_1 + V_2 + \cdots + V_n \tag{5}$$

In the above expression, A_k represents the area of each opening.

It can be seen from equation (5) that the sum of the flow velocity at each opening is proportional to the flow at the inflow boundary, and the proportional relationship is the ratio of the area of the inflow boundary of the cavity fluid domain to the area of each opening. However, from the actual engineering experience and the flow velocity cloud map of the fluid domain completed by the research group using Fluent simulation (see Figure 3) and the mass flow results of each opening in the figure (see Table 1), it can be seen that the flow rate of each opening is not evenly distributed, and when the spacing of the openings is different, the flow rate of each opening will also be different.

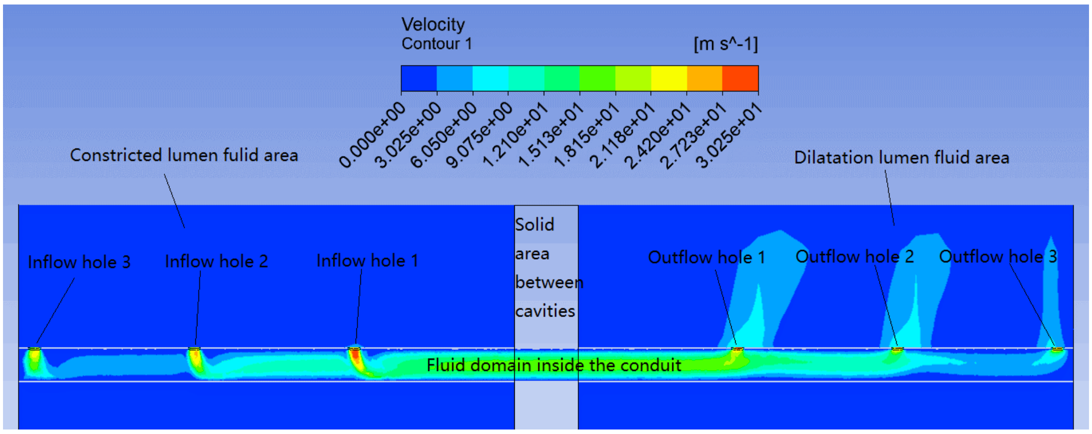

Fig. 3 The velocity cloud diagram of the fluid domain at the cross section of the duct

Table 1 Figure 3 mass flow rate of each opening under different cavity contraction speed

Cavity shrinking and expanding speed $v/(\mathrm{m \cdot s^{-1}})$	Inflow orifice mass flow $Q/(\mathrm{kg \cdot s^{-1}})$			Outflow orifice mass flow $Q/(\mathrm{kg \cdot s^{-1}})$		
-	Inflow orifice 1	Inflow orifice 2	Inflow orifice 3	Outflow orifice 1	Outflow orifice 2	Outflow orifice 3
0.1	0.218	0.239	0.289	0.252	0.244	0.249
0.2	0.443	0.482	0.564	0.482	0.486	0.520
0.3	0.661	0.721	0.838	0.713	0.724	0.781
0.4	0.875	0.958	1.109	0.943	0.989	1.037
0.5	1.089	1.190	1.373	1.169	1.187	1.294

In order to further understand the flow ratio relationship between the inflow and outflow holes on the conduit under the condition of different spacings between the openings, a Fluent simulation model will be established and relevant experiments will be designed to simulate the working state of the device and record the openings. The mass flow rate of each opening is analyzed, and the flow distribution law of each opening is analyzed.

ISSN: 2664-9640

3. Model parameters and test methods

3.1. Model design

The fluid domain models of the cavities inside the conduit and the cavities outside the conduit were established using Solidworks, the cross-sectional schematic diagram of the 3D model of the fluid domain is shown in Figure 4.

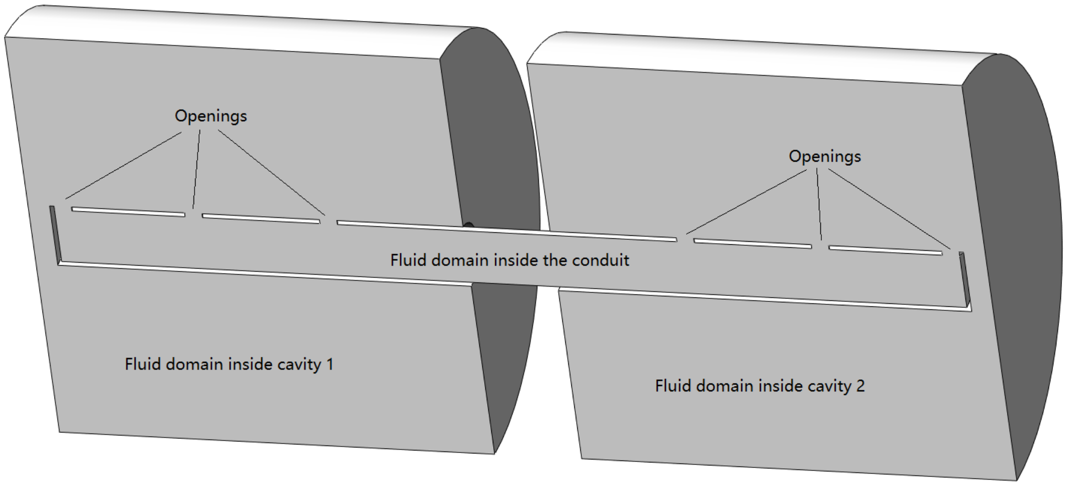

Fig. 4 Schematic diagram of the cross-section of the fluid domain model inside and outside the pipe fitting

The catheter model is provided with 6 openings symmetrically distributed on the left and right sides for the inflow and outflow of fluid, and a cylindrical fluid area is set on each of the left and right ends outside the catheter model, which are Cavity 1 and Cavity 2 respectively. internal watershed. Based on the actual working conditions of the device and the calculation characteristics of the Fluent fluid simulation software [4], the two opposite sides of the two cylindrical regions are set as the inflow and outflow boundaries of the fluid, respectively. The cross-sectional details of the fluid domain structure are shown in Figure 5. Show.

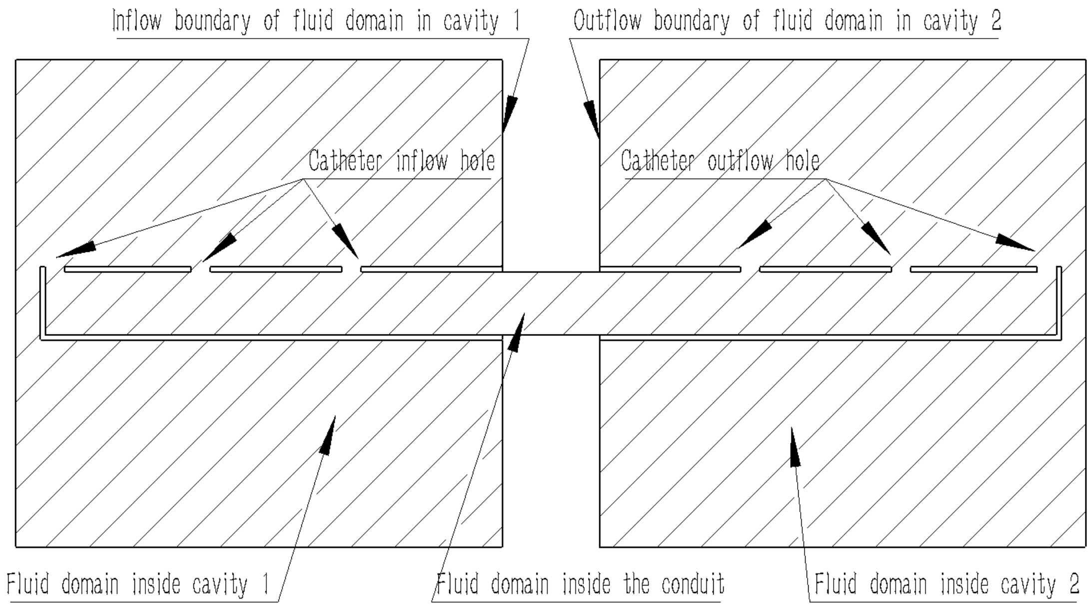

Fig. 5 Detailed cross-sectional view of the fluid domain

In the entire fluid domain simulation process, the fluid first flows into the fluid area in the left cavity 1 of the conduit from the inflow boundary of the cavity 1 at a set speed, and the fluid

ISSN: 2664-9640

entering the cavity 1 passes through the inflow hole of the conduit Enter the inner fluid domain of the catheter, and then the fluid enters the fluid area in the cavity 2 on the right side of the catheter from the outflow hole of the catheter, and finally flows out from the outflow boundary of the fluid domain of the cavity 2. Different calculation models only change the spacing of each opening, and other calculation parameters remain the same.

3.2. Parameter setting and test method

The model parameters formulated according to the above-mentioned fluid domain model are shown in Table 2. Among them, the spacing of the openings at both ends of the conduit is set in turn according to the distance between each opening and the inflow and outflow boundary of the fluid domain in the outer cavity. they are 50mm, 100mm, 150mm equidistant arrangement and 30mm, 80mm, 150mm and 70mm, 120mm, 150mm variable pitch arrangement respectively. According to the actual working environment where the catheter is located, the fluid density of each fluid domain is set to 970kg/m3, the dynamic viscosity is 0.81Pa·s, and the wall friction force of the fluid domain is set according to the default parameters [5].

Table 2 Setting parameters of pipe fittings and fluid domain

Catheter length	Outer diameter of catheter	Inner diameter of catheter	Outer fluid domain diameter	Outer fluid domain length	Opening diameter
330mm	11mm	10mm	100mm	160mm	4mm

When the Fluent fluid simulation is performed, the fluid domains of the above three types of opening spacing are measured at the inflow boundary velocities of 0.1 m/s, 0.2 m/s, 0.3 m/s, 0.4 m/s and 0.5 m/s and calculate the proportion of the inflow and outflow of each opening to the total outflow and inflow of the fluid domain in the conduit [6]. In order to record the total inflow and outflow of the conduit and the flow data of each opening, the left and right three flow inflow holes and flow outflow holes of each spacing type are recorded as the relative positions shown in Figure 6 as Q_1, Q_2, Q_3 and Q_1', Q_2', Q_3'.

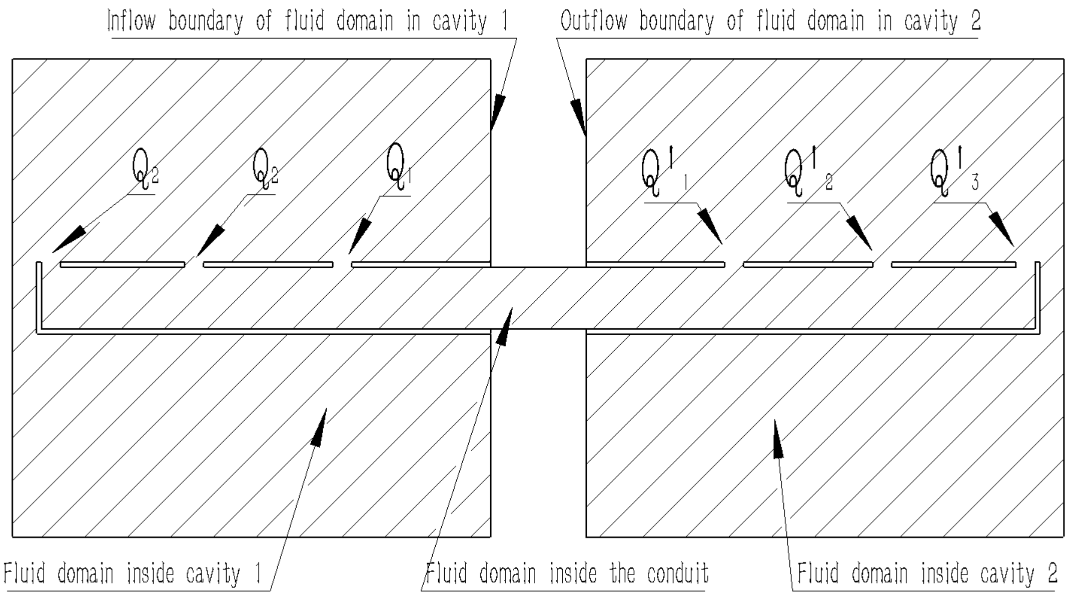

Fig. 6 Schematic diagram of marking of each opening

ISSN: 2664-9640

4. Simulation results and data processing

4.1. Analysis of flow proportioning of conduit inflow holes

Firstly, the flow distribution ratio of each inflow hole on the pipe fitting is analyzed, and the total inflow of the fluid domain in the conduit and the inflow flow of each inflow hole are recorded according to the flux report simulated by Fluent [7], and then each inflow hole is calculated according to the obtained data. The proportion of the flow rate in the total inflow is shown in Table 3 for the recorded flow data and the proportioning relationship.

Table 3 The total inlet flow rate of the conduit and the flow rate distribution ratio of each inlet hole

Spacing type/mm	total inflow Q/(kg·s^{-1})	Traffic distribution ratio		
-	Q_{ai}	Q_1/Q_{ai}	Q_2/Q_{ai}	Q_3/Q_{ai}
50, 100, 150	0.746	0.387	0.321	0.292
50, 100, 150	1.489	0.379	0.324	0.297
50, 100, 150	2.220	0.377	0.325	0.298
50, 100, 150	2.942	0.378	0.325	0.297
50, 100, 150	3.653	0.376	0.326	0.298
30, 80, 150	0.746	0.396	0.324	0.280
30, 80, 150	1.485	0.384	0.327	0.289
30, 80, 150	2.189	0.381	0.329	0.290
30, 80, 150	2.874	0.379	0.331	0.290
30, 80, 150	3.542	0.377	0.333	0.290
70, 120, 150	0.746	0.383	0.315	0.302
70, 120, 150	1.488	0.376	0.320	0.304
70, 120, 150	2.218	0.374	0.323	0.303
70, 120, 150	2.941	0.373	0.324	0.303
70, 120, 150	3.654	0.372	0.325	0.303

Taking the distance between each inflow hole and the inflow boundary as the abscissa, and the flow ratio of each opening as the ordinate, the scatter diagram is shown in Figure 7.

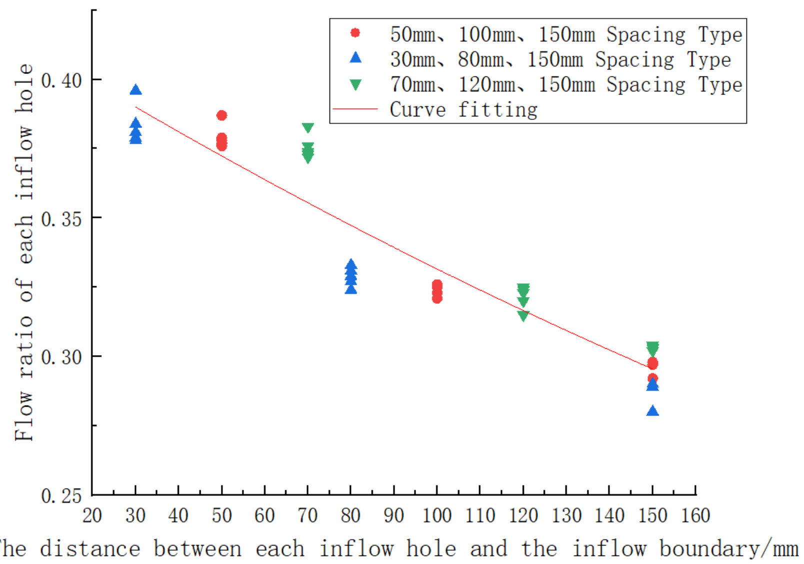

Fig. 7 Scatter plot of flow distribution ratio of inflow holes

ISSN: 2664-9640

The quadratic curve equation fitted from the scatter plot using the regression algorithm is

$$y = (9E - 7)x^2 - 0.001x + 0.4177 \tag{6}$$

The fitting results show that the R^2 value of the curve equation is 0.8967. In order to verify the reliability of the equation fitted by the regression algorithm in the figure, the same opening spacing was substituted into the obtained curve equation, and the corresponding results and calculation errors were calculated as shown in Table 4.

Table 4 Verification results and errors of the inflow hole of the catheter

Spacing type/mm	Regression algorithm verification results and errors%					
-	Q_1/Q_{ai}	error	Q_2/Q_{ai}	error	Q_3/Q_{ai}	error
50, 100, 150	0.370	4.39	0.327	1.83	0.288	2.43
50, 100, 150	0.370	2.43	0.327	0.92	0.288	3.12
50, 100, 150	0.370	1.89	0.327	0.61	0.288	3.47
50, 100, 150	0.370	2.16	0.327	0.61	0.288	3.12
50, 100, 150	0.370	1.62	0.327	0.31	0.288	3.47
30, 80, 150	0.389	1.80	0.343	5.54	0.288	2.78
30, 80, 150	0.389	1.29	0.343	4.89	0.288	0.35
30, 80, 150	0.389	2.06	0.343	4.08	0.288	0.69
30, 80, 150	0.389	2.57	0.343	3.50	0.288	0.69
30, 80, 150	0.389	3.08	0.343	2.92	0.288	0.69
70, 120, 150	0.352	8.09	0.311	1.27	0.288	4.86
70, 120, 150	0.352	6.38	0.311	2.81	0.288	5.56
70, 120, 150	0.352	5.88	0.311	3.72	0.288	5.21
70, 120, 150	0.352	5.63	0.311	4.01	0.288	5.21
70, 120, 150	0.352	5.38	0.311	4.31	0.288	5.21

4.2. Analysis of flow proportioning of conduit outflow holes

The ratio analysis of each outflow hole on the conduit is the same as the analysis method of the inflow hole. It also records the total outflow of the watershed in the pipe fitting and the outflow of each outflow hole according to the flux report simulated by Fluent, and then calculates each outflow according to the obtained data. The flow rate of the orifice accounts for the proportion of the total outflow. The flow data and ratio relationship of the conduit and each opening recorded by Fluent simulation are shown in Table 5.

Table 5 The total flow rate of the conduit and the flow rate distribution ratio of each outflow hole

Spacing type/mm	total inflow Q/(kg·s^{-1})	Traffic distribution ratio		
-	Q_{ao}	Q_1/Q_{ao}	Q_2/Q_{ao}	Q_3/Q_{ao}
50, 100, 150	0.746	0.339	0.327	0.334
50, 100, 150	1.488	0.324	0.327	0.349
50, 100, 150	2.219	0.321	0.326	0.352
50, 100, 150	2.939	0.321	0.326	0.353
50, 100, 150	3.651	0.320	0.325	0.355
30, 80, 150	0.746	0.340	0.335	0.325
30, 80, 150	1.485	0.324	0.335	0.341

30, 80, 150	2.188	0.321	0.334	0.345
30, 80, 150	2.875	0.320	0.332	0.348
30, 80, 150	3.538	0.319	0.332	0.349
70, 120, 150	0.746	0.339	0.325	0.336
70, 120, 150	1.488	0.325	0.326	0.349
70, 120, 150	2.216	0.321	0.326	0.353
70, 120, 150	2.939	0.320	0.326	0.354
70, 120, 150	3.650	0.320	0.326	0.354

The scatter diagram of the flow distribution ratio of each outflow hole is shown in Figure 8.

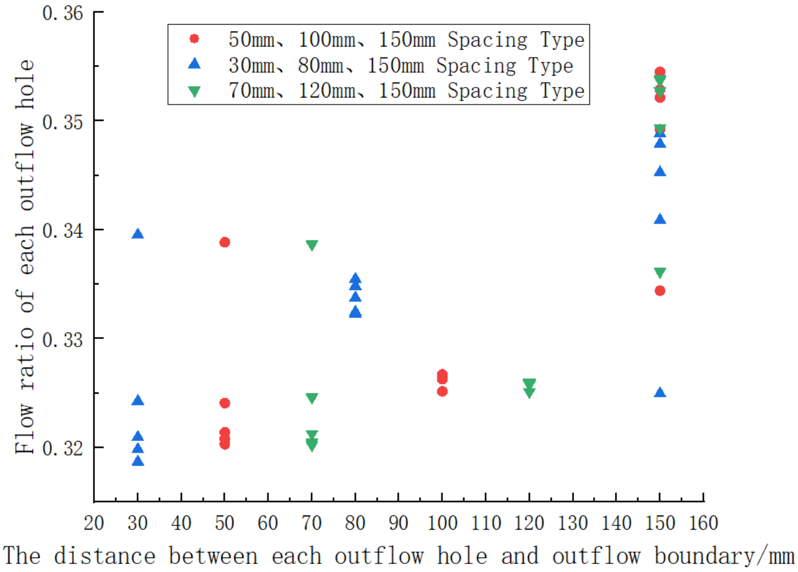

Fig. 8 Scatter diagram of flow distribution ratio of outflow orifice

Compared with the inflow orifice flow distribution scattergram, the data points in the outflow orifice flow distribution scattergram are more scattered, and it is no longer applicable to find the flow distribution law by curve fitting. In terms of the numerical value in the comprehensive table and the distribution range of each data, each data point hovers around 0.33, so you can try to calculate the error according to the average of the total outflow, that is, one-third of the total flow. , the calculation results and errors are shown in Table 6.

Table 6 Verification results and errors of catheter outflow holes

Spacing type/mm	Total outgoing traffic $Q/(kg\cdot s^{-1})$	Calculate the error as one-third of the total outflow%		
-	Q_{ao}	Q_1'/Q_{ao}	Q_2'/Q_{ao}	Q_3'/Q_{ao}
50, 100, 150	0.746	1.63	1.98	0.32
50, 100, 150	1.488	2.77	2.00	4.55
50, 100, 150	2.219	3.58	2.06	5.34
50, 100, 150	2.939	3.76	2.11	5.34
50, 100, 150	3.651	3.90	2.45	5.97
30, 80, 150	0.746	1.83	0.64	2.51
30, 80, 150	1.485	2.72	0.45	2.22
30, 80, 150	2.188	3.71	0.13	3.46
30, 80, 150	2.875	4.04	0.32	4.18

30, 80, 150	3.538	4.39	0.27	4.45
70, 120, 150	0.746	1.58	2.46	0.85
70, 120, 150	1.488	2.61	2.19	4.58
70, 120, 150	2.216	3.62	2.21	5.50
70, 120, 150	2.939	3.92	2.24	5.81
70, 120, 150	3.650	3.86	2.26	5.76

From the data in Table 6, it can be seen that the maximum error of the flow rate of each outflow hole calculated by one third of the total outflow rate does not exceed 5.97%, so the flow rate of the outflow hole on the conduit can be regarded as a uniform distribution, which shows that it is more reasonable to calculate the average flow rate according to the number of outflow holes.

5. Summary

In this paper, a three-dimensional simulation model of the fluid domain inside and outside the conduit is established to study the flow distribution law of each opening of the porous conduit, and the flow data of each opening is obtained through the Fluent fluid simulation. Two conclusions:

1) According to the results obtained from the simulation, the inflow holes on the porous conduit follow a certain curve relationship with the distance from the inflow boundary, and the flow rate of the outflow holes is basically the total outflow flow, which is evenly distributed according to the number of outflow holes. The results obtained in this test have certain guiding significance for the design of porous pipe fittings with other spacing types. When porous pipe fittings with other spacing types are required, the preliminary energy efficiency calculation can be performed with reference to the curve equation obtained in this test.

2) The Fluent fluid simulation scheme designed in this experiment is also applicable to other porous pipes with two or more than three ends. For other porous pipes with different numbers of holes and different distances between holes, the flow of each hole is studied. The simulation test method described in this paper can still be used for research. Fluent fluid simulation software has strong advantages, especially when studying the flow laws of such confined spaces and it is difficult to design actual specimens for flow monitoring.

References

[1] Datong Qin, Liyang Xie. Modern Machinery Design Manual[M]. Beijing: Chemical Industry Press, 2011

[2] Gonghui ZHANG, Zhihang LI, Zhihong ZHOU, et al. Air-passage structure improving of pneumatic electromagnetic valve based on flow field simulation withing Fluent[J]. Hydraulics Pneumatics & Seals, 2010, 30(12):41-43 (in Chinese)

[3] Sijing Wang, Shaomin Peng, Yongjiu Shi, et al. Engineering Fluid Mechanics[M]. Wuhan: Huazhong University of Science and Technology Press, 2006

[4] Zhanzhong Han, Jing Wang, Xiaoping Lan, et al. Fluent fluid engineering simulation calculation example and application[M]. Beijing: Beijing Institute of Technology Press, 2010

[5] Jun Xie, Hongliu Guo, Wei Chen, et al. The flow field analysis based on ANSYS and Fluent software in honeycomb duct[J]. Machinery Design & Manufacture 2008,(09):70-72 (in Chinese)

[6] Chunfang Xu, Jiandong Gao, Daiqiao Zhang, et al. Experimental Research on Multi-Outlet Flow Distribution of Low-Pressure Irrigation Conduit. Journal of Irrigation and Drainage, 2020,39(S1):96-100.

[7] Kun Hu, Tingting Hu, Haifeng Ma, et al. ANSYS Fluent Detailed Example. Beijing: Machinery Industry Press, 2018