Research progress and numerical simulation of pressure compensating emitter

Min Xiao
Jiangxi Water Resources Institute, Nanchang 330013, Jiangxi, China
*Corresponding author’s e-mail: Min Xiao, Senior Engineer; research interests: theory and technology of water conservancy and hydropower engineering. e-mail: 948384429@qq.com.

Abstract: The research status of the emitter at home and abroad is described. By reading the relevant literature, the feasibility of the numerical simulation analysis method in the study of the internal flow analysis of the emitter and the optimization design of the emitter structure is obtained. Compared with other emitters, the pressure compensating emitter has the advantages of free flow from pressure change, good anti-plugging performance and self-flushing function. The method of combining numerical simulation with PIV (particle image velocimetry) can be used to study it and optimize the hydraulic performance and anti-plugging performance of the pressure compensating emitter.

1. Introduction
China is a country with an overall shortage of water resources. The annual average precipitation is about 6.188 trillion m³, and the annual water resources is about 2.7 trillion m³, ranking the sixth place in the world, but the per capita total is only 2,250m³, only 1/4 of the world average level. The water consumption of agricultural irrigation accounts for more than 80% of the total water consumption of the national economy. Drought and water shortage have become the main constraints of agricultural development in China. The utilization rate of irrigation water in China is only 40%, which is far behind that in developed countries. It can be seen that the development of agricultural water-saving irrigation is very important and the potential of water-saving is also great. Drip irrigation is one of the best micro irrigation techniques with simple principle. It mainly drips slowly to the soil through the main pipe, branch pipe, capillary and emitter on the capillary under the condition of low pressure. It is an irrigation system that directly supplies filtered water, fertilizer and other chemicals to the soil. Drip irrigation technology is the highest water-saving irrigation technology among many water-saving irrigation technologies in the world, with water-saving efficiency of more than 90%; it can also realize the simultaneous supply of water and fertilizer, the precise control of water and fertilizer, and improve the utilization ratio of water and fertilizer, so it is widely used in the world. One of the characteristics of the drip irrigation system is the small water flow of the emitter, generally the flow of a single emitter is 1-10l/h, the water outlet of the emitter is very small, generally only about 0.5-1.2 mm, so the emitter is easily blocked by the dirt in the water; another outstanding feature of drip irrigation is the high uniformity of drip irrigation system. The uniformity of the system is mainly affected by the hydraulic performance of the emitter, in addition to the operating state of the system. Based on the above two characteristics of the drip irrigation system, the performance of the emitter (anti plugging performance, hydraulic performance) directly affects the operation effect of the drip irrigation system,
and the emitter becomes the key component to determine the performance and success of the drip irrigation system.

2. **Research status at home and abroad and numerical simulation**

2.1. **Research status of emitter at home and abroad**

The flow passage of the emitter is narrow, and blockage is very common in field application. The plugging factors of drip irrigation system can be divided into physical, chemical and biological factors [1]. Domestic professional researchers have carried out experimental research on these three plugging mechanisms. Yan Dazhuang, Yang Peiling et al. conducted comparative experiments with clear water and muddy water of different concentrations, and finally came to the conclusion that the flow rate of emitter first increased and then decreased with the increase of sand concentration, while the flow index of emitter slightly decreased [2]. Liu Haijun and Huang Guanhua et al. studied the flow rate, irrigation uniformity and blockage of three kinds of emitters (single wing labyrinth, inlaid emitter and pressure compensation hole emitter) respectively, using reclaimed water and tap water as water sources. The results show that the flow rate and uniformity of single wing labyrinth emitter decrease the most, and the change of pressure compensation hole emitter is the least, and it is pointed out that chemical precipitation (mainly CaCO$_3$ and MgCO$_3$) is the main way to cause emitter blockage [3]. Xue Yingwen pointed out that the organic and microorganism in the water will cause the blockage in the emitter, but the inorganic particles are the main cause of the blockage.

Good or bad water quality is an important reason for the difficult or easy formation of the blockage in the emitter, and the internal structural parameters of the emitter will also affect the anti-plugging performance of the emitter. Domestic professional researchers have also made relevant research on the influence of the structure parameters of the emitter on the hydraulic and anti-plugging performance of the emitter. Li Guangyong, et al. set up a comprehensive test device for the hydraulic and anti-plugging performance of indoor micro emitters in China, and tested the hydraulic and anti-plugging performance of 16 kinds of emitters designed and manufactured. The results showed that the angle and height of I teeth in the flow passage pitch had a great influence on the anti-plugging performance of emitters, while the depth of teeth had little influence [4]. Through the experimental research on the anti-plugging performance of 15 kinds of emitters, Mu Naijun, Zhang Xin, et al. obtained that there are limitations in characterizing the anti-plugging performance of emitters by a single structural parameter (flow passage width W, flow passage depth D, tooth height h and tooth angle), while the structural characteristic parameters (minimum length of section min (D, W), hydraulic radius R and tooth spacing I) can better reflect the characteristics of the cross section and vertical section of emitters [5]. Wang Jiandong et al. conducted orthogonal experiments with different tooth spacing, tooth angle, tooth height and flow passage depth, and concluded that the reason for poor anti-plugging performance of large flow emitter can be attributed to its smaller flow passage width. Li Yunkai, Yang Peiling, et al. measured the geometric parameters of the labyrinth flow passage of the emitter by combining AutoCAD and reading microscope, constructed the plane model of the cylinder emitter by using the stainless steel spark wire technology, visualized the flow field inside the emitter, and provided a new idea for the development of the emitter [6].

2.2. **Numerical simulation method**

The structure parameters of the emitter have a very important impact on the hydraulic performance and anti-plugging performance of the emitter. Through the in-depth study of experts at home and abroad, the hydraulic and structural performance of the emitter has also been greatly improved. However, due to the small size and complex structure of flow passage of the emitter, the current flow measurement technology is very difficult to observe the internal flow field of the emitter, so the relationship between the internal flow field and the working performance of the emitter is still lack of deep understanding and grasp. In recent years, computational fluid dynamics (CFD) has developed rapidly. It can calculate the velocity field and pressure field of fluid in three-dimensional area under
complex boundary conditions, as well as the hydraulic loss, pressure fluctuation, turbulence and other information in the flow passage by numerical simulation. Because the numerical simulation based on CFD can better understand the flow regularity and reduce the test period, so now some people have applied CFD technology to the theoretical analysis, model establishment and parameter selection of the emitter.

The numerical analysis of the internal flow passage of the emitter began in the late 1990s. The basic idea of this research method is [7]: firstly, the mathematical model of the flow field of the emitter, i.e. the control equation in the form of partial differential equation, is established by using the theory of computational fluid dynamics. Then the governing equations are transformed into algebraic equations on discrete nodes by discrete numerical methods (finite element method, finite volume method). Finally, the algebraic equations are solved to obtain the velocity and pressure at the calculated nodes in the flow field. The advantage of this method is that as long as the computational grid is fine enough, the three-dimensional velocity, pressure and temperature of any point in space can be calculated theoretically. So far, the CFD analysis of the flow field in the emitter is still in its infancy. Liu Xiaomin, Wang Shangjin, Meng Guixiang, et al. carried on the numerical calculation to the cylinder labyrinth emitter, analyzed the flow characteristic of the labyrinth emitter under the different flow rate. Wei Qingsong et al. conducted analogue simulation of the flow field in the emitter by using ANSYS and visualized the hydraulic characteristics of the emitter. Salvador et al.[8] of Purdue University in the United States established a three-dimensional model with PROE, used laminar flow model, and used commercial software FLUENT to analyze the hydraulic characteristics. It is believed that the numerical simulation can accurately simulate the pressure change value of labyrinth emitter. Li Yongxin et al. used CFD to simulate the pressure flow relationship of the emitter, the internal pressure and velocity distribution of the flow passage, and obtained that the simulation and the measured value have good consistency, and CFD numerical simulation can provide an effective research means for further research on the hydraulic performance of the emitter. Yan Dazhuang et al. used CFD technology to digitally simulate the movement of suspended particles with different concentrations in the labyrinth emitter flow passage, and used two-phase flow model to analyze the fluidity of suspended particles. The results show that the simulation results of DPM model for the distribution characteristics of suspended particles are in good agreement with the existing experimental data, which can be used as a means of simulation. Taking the inlaid chip type emitter and cylindrical emitter as the object, Liu Haisheng et al. [9] adopted the way of combining DPIV technology and large eddy simulation for the first time to study the flow characteristics in the flow passage, It was observed that the flow movement in the flow passage can be divided into the main flow area and the non-main flow area by using DPIV technology, in which the movement speed of the non-main flow area is relatively small, and the solid particles in the water are easy to precipitate in this area, resulting in blockage.

3. Pressure compensating emitter and numerical simulation

At present, most of the design methods of the emitter are mainly to reduce the water pressure by using the long flow passage. But because the size of the emitter is too small, in order to form a long flow passage, it is necessary to reduce the cross-sectional area of the flow passage, which is an important reason for the poor anti plugging performance of the emitter. In addition, the most widely used labyrinth flow passage emitter at present is suitable for fixed flow. When the flow changes in a large range, especially it is not suitable under the condition of large slope terrain in the northwest of China, which has a certain impact on the promotion of efficient water-saving agriculture. The pressure compensating emitter has many advantages, such as the flow of water is not affected by the change of pressure, good anti plugging performance and self-flushing function, which is gradually recognized by people.

According to the different connection ways with the drip irrigation pipe, there are mainly two types: on pipe type and inlay type. The on pipe pressure compensating emitter has the characteristics of
convenient disassembly, etc. The working principle of on pipe pressure compensating emitter is introduced below.

3.1. The working principle of pressure compensating emitter

There is an elastic diaphragm in the adjusting chamber, and a flow passage and a water outlet are arranged under the diaphragm. When the emitter is in the compensation state, the cross-sectional area of the flow passage between the elastic diaphragm and the base will change with the change of the pressure, the pressure increases, the cross-sectional area of the flow passage decreases, the pressure decreases, and the cross-sectional area of the flow passage increases, so that the flow rate of the emitter remains stable; when the pressure is reduced to a certain extent, the emitter will get into the flushing state, the elastic diaphragm will be completely separated from the base, the outlet flow passage will become larger, and the flow rate will increase accordingly.

The working process of the pressure compensating emitter is as follows: After the water enters from the water inlet, it directly enters the upper part of the elastic diaphragm in the compensation area, thus forming a pressure difference between the upper and lower parts of the elastic diaphragm, which causes the elastic diaphragm to deform, and the inlet pressure is different. When a certain proportion relationship is formed between the inlet pressure and the flow cross-section area, the flow rate may be kept constant.

Assuming that the minimum flow cross-section area of the compensation area is \( \omega \), then:

\[
\omega = a \times b
\]  

(1)

In the formula: 
- \( a \) - depth of compensation area, m;
- \( b \) - minimum flow cross-section width of the compensation area, m, when the compensation area is circular, \( b \) is the diameter of the compensation area; when the compensation area is rectangular, \( b \) is the width of the compensation area.

Due to the effect of the uniform pressure difference \( \Delta P \) on the elastic diaphragm, the reduced cross-sectional area after deformation is \( \omega_b \), and the cross-sectional area of the flow passage of the emitter after deformation is \( A = \omega - \omega_b \). After getting \( \omega_b \), \( A \) can be obtained. According to the engineering mechanics, the deformation of the elastic diaphragm under the action of \( \Delta P \) should be circular arc, as shown in the following figure:

![Diagrammatic sketch of elastic diaphragm deformation](fig1.png)

It can be seen from the figure that the arc area:

\[
\omega_b = \frac{1}{2} [r^2 \theta - b \sqrt{r^2 - \left(\frac{b}{2}\right)^2}]
\]  

(2)

And \( b = 2r \sin \frac{\theta}{2} \), arc length is \( L = r\theta \), we can get:

\[
\omega_b = \frac{1}{2} [rL - b \sqrt{r^2 - \left(\frac{b}{2}\right)^2}]
\]  

(3)

In the formula, \( r \) - radius of circular arc after deformation of elastic diaphragm, m; \( \theta \) - central angle of circular arc after deformation of elastic diaphragm;
From the mechanics point of view, the deformation of the elastic diaphragm is due to the axial force stretching. Assume this force as $N$, take half of the diaphragm as the mechanics analysis object, as shown in the following figure:

![Fig 2. Isolated body](image)

After deformation and stability, the elastic diaphragm should be in a balanced state. From that, the resultant force in the direction of the force on the diaphragm is zero, we can get:

$$\Delta P \times \frac{b}{2} = N \cos(90° - \frac{\theta}{2}) = N \sin\frac{\theta}{2}$$

i.e.

$$N = \frac{b \Delta P}{2 \sin\frac{\theta}{2}}$$

(4)

(5)

It is known from engineering mechanics that the axial force makes the elastic diaphragm elongate:

$$\frac{N}{e} = E \left(\frac{L - b}{b}\right) = E \left(\frac{L}{b} - 1\right)$$

In the formula, $E$ - modulus of elasticity, MPa;
$e$ - thickness of elastic diaphragm, mm.

From the formula (5) and (6), we can get:

$$L = b \left(\frac{N}{Ee} + 1\right) = b \left[\left(\frac{b \Delta P}{2 Ee \sin\frac{\theta}{2}}\right) + 1\right] = b \left(\frac{r \Delta P}{Ee} + 1\right)$$

(7)

The flow rate of the emitter depends on two factors: the flow rate and the cross-sectional area of the flow passage, while the flow rate is related to the pressure, as shown in the following formula:

$$Q = V \omega = \sqrt{\frac{2g \Delta P}{\gamma}} (ab - \omega_i)$$

(8)

The formula (4), (5) and (7) are combined to get the flow relation formula:

$$Q = \sqrt{\frac{2g \Delta P}{\gamma}} \left[\frac{1}{2} \left(\frac{r \Delta P}{\omega E} + 1\right) - b \sqrt{r^2 - \left(\frac{b}{2}\right)^2}\right]$$

(9)

In the formula, $Q$ - emitter flow, L·h$^{-1}$
$V$ - flow rate of the minimum flow cross-section area of the compensation area, m·s$^{-1}$
$\gamma$ - Volume weight of pressure water, N·m$^{-3}$

It can be seen that the factors that affect the flow rate of the pressure compensating emitter are: the pressure difference between the upper and lower part of the elastic diaphragm, the width of the compensation area, the depth of the section, the radius of the circular arc after the deformation of the elastic diaphragm, the thickness of the elastic diaphragm, and the modulus of elasticity.

3.2. CFD simulation of flow field in pressure compensating emitter

At present, the simulation and development of the new type of emitter is usually after the completion of the structural design, the injection molding of the physical products, and then the assembly of the outer tube in the production line. Finally, it is connected to the emitter hydraulic performance test...
device to measure the relationship between its flow and pressure. The whole process has a long cycle and high cost. If the test results do not meet the design requirements, it is necessary to return to modify the structure of the emitter and the corresponding meld structure, resulting in a great waste. Through the numerical simulation test based on CFD, we can fully understand the flow regularity, greatly reduce the workload of laboratory and test entity test research, shorten the development cycle and reduce the cost. Commonly used software to calculate fluid structure coupling are: ANSYS, ADANA, FLUENT, etc. In the following, a method based on CFD (fluent) is proposed to analyze the flow passage and flow field of pressure compensating emitter.

3.3. CFD simulation of flow field in pressure compensating emitter

The size of the flow passage of the pressure compensating emitter is generally between 0.3-2 mm. It can be considered that the fluid flowing in the passage is continuous, steady and incompressible. Because of the complex structure of the internal flow passage and the constantly changing boundary, it is considered as a disturbance to the flow movement, so it is not appropriate to use the Reynolds number 2320 of the smooth circular tube to divide the water state. Li Yunkai, Yang Peiling et al. think that RE in labyrinth flow passage is between 105-930, so the critical Reynolds number of flow transition is smaller than that of conventional scale flow passage, which is lower than 255. The flow field in the pressure compensating emitter can be regarded as turbulent flow. Therefore, the continuity equation, momentum equation and energy equation can be used for the governing equation. However, from the existing data, the flow regularity of the micro flow passage is different from the conventional scale. After considering the influence factors in a certain range, the flow process can be described by the N-S equation. Due to the effect of the liquid surface tension and viscous force, the research is more complex [10,11].

3.4. CFD pre-processing

With the help of electron microscope, we can measure the size of the inner flow passage of the cylindrical pressure compensating emitter, and use the three-dimensional software PRO/E or NX to build the model of the emitter flow passage.

3.5. CFD post-processing

The post-processing program in FLUENT is also available in TECPLOT. It is mainly to conduct microscopic analyze in the flow field of the emitter, which can get the velocity field, pressure field and parameter analysis on a certain section conveniently and accurately. The conclusion is obtained by analyzing the numerical distribution on these planes.

3.6. PIV particle image measuring and testing technology

Particle image measuring and testing technology is a kind of optical velocity measurement technology, which can obtain the whole flow information in a certain instant in the field of view. The working principle is to measure the average velocity of particles by measuring the moving distance of tracer particles at a certain time interval.

4. Conclusions and suggestions

(1) The plugging of emitter will lead to reducing the uniformity of emitter irrigation, poor quality of irrigation, shortening the service life, and even making the emitter system unusable. At present, the plugging of emitter is the main problem in the application of drip irrigation system. Professional researchers at home and abroad have made relevant research on the anti-plugging performance of the emitter.

(2) Based on the numerical simulation method of computational fluid dynamics (CFD), the complex flow field in the emitter is visualized. The velocity field and pressure field of the fluid in the three-dimensional region, as well as the hydraulic loss, pressure fluctuation and turbulence in the flow.
passage are calculated by numerical simulation. Based on the above advantages, the numerical simulation method is accepted by more and more people.

(3) The pressure compensating emitter has many advantages, such as no influence of pressure change, good anti plugging performance and self-flushing function. However, from the existing data, the research on the numerical simulation of emitter flow passage mainly focuses on various labyrinth emitters.

(4) The CFD numerical simulation method can be used to try to simulate the flow field in the pressure compensating emitter, and PIV particle test technology can be used to verify. Through the analysis of the calculation results, the characteristic parameters of the internal flow passage of the pressure compensating emitter can be reasonably designed and optimized.

References
[1] Bucks, D. A., Nakayama, F. S., Gilber, R.G. (1979) Trickle water quality and preventive maintenance. Agricultural Water Management, 2: 149-162.
[2] Yan Dazhuang, Yang Peiling, Zhao Qiao. (2008) Experimental study on sediment flow hydraulic characteristics in dripper emitter path. Transactions of the CSAE, 24(2): 52-55.
[3] Liu Haijun, Huang Guanhua, Wang Pengchao, et al. (2009) Effect of drip irrigation with reclaimed water on emitter clogging. Transactions of the CSAE, 2009, 25(9): 15-20.
[4] Li Guangyong, Wang Jandong, Alam, M., et al. (2006) Influence of geometrical parameters of labyrinth flow path of drip emitters on hydraulic and Anti-clogging performance. ASABE Paper, 49(3): 637-643.
[5] Mu Naijun, Zhang Xin, Li Guangyong, et al. (2007) Experimental study on anti-clogging performance of dental labyrinth flow passage of drip emitters. Transactions of the CSAE, 23(8): 34-39.
[6] Li Yunkai Yang Peiling, Ren Shumei, et al. (2006) Plane Model of Labyrinth Path Drip Irrigation Column Emitter. Transactions of the CSAE, 37(4): 48-51.
[7] Collins John, Tan Yung-chieh. (2003) Lee Abraham P. Optimization of shear driven droplet generation in a microfluidic device. ASME, 11:15-21.
[8] Salvador, P. G., Arviza, Valverde J., Bralts V. F. (2004) Hydraulic flow behaviour through an in-line emitter labyrinth using CFD techniques. ASAE Paper, 45(3): 627-633.
[9] Liu Haisheng, Li Yunkai, Liu Yanzheng, et al. (2010) Flow characteristics in energy dissipation units of labyrinth path in the drip irrigation emitters with dpiv technology. Journal of Hydrodynamics, 22(1):137-145.
[10] Ohiuddin, M., Mala, G., Li Dongqing. (1999) Flow characteristics of water in microtubes. Heat and Fluid Flow, 20: 142-148.
[11] Thorsen, T., Richard W. (2001) Dynamic pattern formation in a vesicle—generating microfluidic device. Physical Review Letters, 86(18): 4163-4166.