E-ISSN 2581 – 7957
P-ISSN 2277 – 3916
CVR Journal of Science and Technology, Volume 24, June 2023
DOI: 10.32377/cvrjst2401

# Hydraulic and Anti-clogging Performance Evaluation of a Novel Cylindrical Drip Emitter using CFD Techniques

Ramamohan Reddy Kasa[1] and Venkata Ramamohan Ramachandrula[2]
[1]Professor, CVR College of Engineering/Civil Engg. Department, Hyderabad, India.
Email: kasarammohan@gmail.com
[2]Executive Director, Water and Livelihoods Foundation, Hyderabad, India.
Email: ramamohan.ramachandrula@gmail.com

***Abstract:*** **An inline cylindrical drip emitter of 4 Lh-1 design flow rate is developed incorporating favourable geometric features against the physical clogging. The emitter model, with one-outlet and two-outlet variants, was tested for its hydraulic and anti-physical-clogging performance using Ansys Fluent 2022 software. The simulated flow rate of 3.988 and 3.992 Lh-1 is found to be very close to the design flow rate. The emitter exponent values of 0.4351 and 0.4406 for the one-outlet and two-outlet variants, respectively, are found to be within the stipulated limit of 0.5 by Bureau of India Standards (BIS) guidelines.**

**The discrete particle modelling (DPM) simulation revealed that 100% of the particles entering the inlets of the emitter variants are escaping within an average residence time of 0.3779-1.036 s for the one-outlet variant and 0.2044-1.005 s for the two-outlet variant. The average residence time increased with increase in the particle size from 0.1 µm to 100 µm, however the same is not affected by the mass flow rate of particles. Overall, the novel cylindrical emitter developed is found to be satisfactory in its hydraulic and anti-physical-clogging performance.**

***Index Terms:*** **CFD, DPM, Ansys fluent, Drip Emitter, Anti-clogging, SST, k-ω Turbulence Model.**

## I. Introduction

Drip irrigation is widely known for its beneficial aspects, such as, improved water use efficiency and uniform application of water, resulting in better water and crop productivity. However, clogging of drip emitters is a major problem that occurs and increases with prolonged use of the drip systems. The emitter clogging affects the emitter flow rate, uniform delivery of irrigation water over the farmland and ultimately, its functional life. The composite clogging of emitters generally involves formation of physical, chemical, and biological deposits. In case of irrigation using groundwater pumped from wells, physical clogging due to the suspended solids (SS) in water and the chemical clogging due to the precipitation of dissolved chemicals (such as, salts of Ca, Mg, Fe etc.) in water are two major concerns.

Conventionally, the emitter clogging is addressed by frequent flushing of lateral pipes; periodic cleaning of disk or screen filter; treatment of laterals with acid for removing the chemical deposits; and chlorination to remove bacteriological clogging. IS 13487, IS 13488 and IS 14791 by Bureau of Indian Standards (BIS) elaborate on the methods of testing emitters, assessing clogging hazards of irrigation water and treatments against clogging (BIS 2015; BIS 2008; BIS 2016).

Drip emitters are primarily of two types, viz., (i) cylindrical and (ii) flat, based on the geometric features. Also, each of them may be classified as (i) regulated and (ii) unregulated emitters, depending on the pressure-flow rate relationship. While irrigation water quality plays a key role in the degree of emitter clogging, the geometric design of emitters is also found to be playing a significant role. Several researchers studied and suggested improved labyrinth path geometries for better anti-clogging performance. In this article, a novel anti-physical-clogging emitter design is proposed, and its performance is analysed using Computational Fluid Dynamics (CFD) techniques.

## II. Literature Review

Recent advances in CFD software opened a new possibility of visualizing the fluid flow as well as particle tracks as they pass through the emitter flow paths. Also, CFD facilitated testing and simulation of hydraulic performance of emitters by varying different geometric features and inlet pressure.

[16] studied labyrinth flow paths of emitters and proposed pressure loss coefficient (PLC) as the index for evaluation of their hydraulic performance. [12] applied standard k-ε and large eddy simulation (LES) methods for CFD simulation of flow through the labyrinth path of a cylindrical drip emitter. The anti-clogging performance of the emitter flow path was studied using the velocity distribution at different sections. [7] analysed the hydraulic and anti-clogging abilities of a flat type drip emitter, using CFD simulations and improved digital particle image velocimetry (DPIV) equipment in the laboratory settings. [10] analysed a trapezoidal labyrinth flow path of a flat emitter, by varying the values of dent spacing and dent angle. It was found that, for a given length of flow path, increase in dent spacing increased the flow rate and increase in dent angle reduced the pressure sensitivity of the emitter. [14] coupled the discrete element method (DEM) with the CFD to model the movement of tiny physical particles of size 65, 100 and 150 µm in the emitter labyrinth path. The study found that larger sized particles are more prone to be caught in the vortex areas of the labyrinth, resulting in eventual settlement, and clogging of the emitter. [2] studied four cylindrical emitter models having circular,


Received on 03-04-2023, Revised on 01-05-2023, Accepted on 11-05-2023.

E-ISSN 2581 – 7957
P-ISSN 2277 – 3916

DOI: 10.32377/cvrjst2401

rectangular, triangular and trapezoidal labyrinth paths and found that triangular one has got the least flow rate and most pressure drop for a given inlet pressure.

CFD modelling not only eased the comparative study of existing emitters, but also facilitated anti-clogging geometric improvements to the emitters. Few researchers optimize the labyrinth flow path of the flat emitters by appropriately altering one or more flow path width, dent angle, dent spacing and boundary features. Few other researchers came up with entirely novel designs of emitters that incorporated new features into the labyrinth paths, such as, divided-flow emitters, pit structure emitters etc.

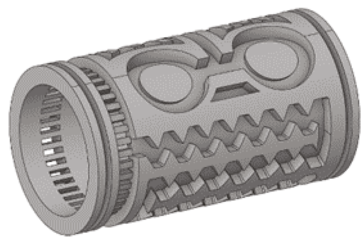
[6] studied a flat type drip emitter with various combinations of dent height (h), flow path width (D) and dent angle (ɵ). Larger dent angle was found to favour pressure insensitivity of the emitters, and boundary optimization helped in improved anti-clogging performance. [1] designed an online anti-clogging drip emitter consisting of connected cones and labyrinth flow path, improving the hydraulic and physical particle flow dynamics through the emitter. [11] designed a divided flow emitter path and optimized its geometric dimensions. The optimal channel had an emitter exponent of 0.515 and a particle passing rate of 88.5%.

[7] optimized the flow path boundary with a curvature of ½ of the flow path width, creating the washing-the-effect leading to improved particle excretion performance. [15] analysed the hydraulic performance and sand particle movement in a labyrinth channel at different dentation angles of 90°, 60°, 45° and 30°. The study recommended dentation angle range of 60° to 90° and a higher velocity level for optimum hydraulic and anti-clogging performance. [9] devised a novel type of stellate water-retaining labyrinth channel structure with optimum emitter exponent value. [13] developed a pit drip irrigation emitter by optimizing the vortex formation areas of the labyrinth path. The flow index and energy dissipation coefficient obtained from CFD simulations were compared with the experimental investigations.

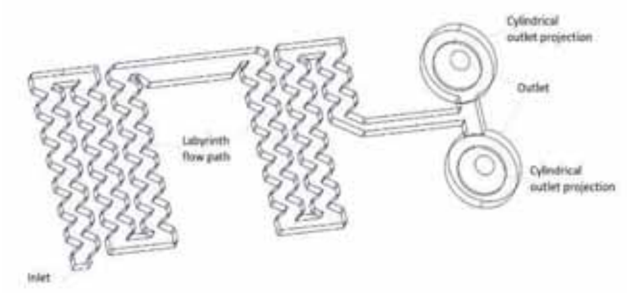
Though most of these works mainly focused on either optimizing the trapezoidal labyrinth path or proposed novel flow path structures, there are none that attempted to optimize the whole emitter geometry. Current work addresses this gap and proposes a novel cylindrical emitter and analyses its hydraulic and anti-physical-clogging performance using CFD techniques.

## III. MATERIALS AND METHODS

A cylindrical emitter model of 4 Lh-1 design flow rate was prepared using AutoCad Fusion 360 software in STEP file format. The emitter model has the option of using one outlet hole or two outlet holes on the lateral pipe. For carrying out Computational Fluid Dynamics (CFD) analysis, the flow path geometries were separated from the emitter models using Ansys SpaceClaim 2022 software (Fig.1).



(a)




(b)

Figure1. CAD Models of (a) Cylindrical Emitter, (b) Unwrapped Flow Path

The performance of the novel emitter model developed was tested in terms of its hydraulic and anti-physical-clogging performance, as described below.

### A. *Hydraulic Performance of Emitter Variants*

Ansys Fluent 2022 student software was used to conduct hydraulic performance studies on the four flow path variants. The flow rate of the emitter models is assessed under steady state flow conditions through the flow paths. Both inlet and outlet(s) are set as pressure boundaries. The two-equation Shear Stress Transport (SST) k-ω turbulent flow model is used for the simulation. The outlet gauge pressure was set as 0 m (= atmospheric pressure). The outlet flow rate measurement was done for the inlet gauge pressure of 100 KPa (10.2 m). In case of two outlet model variant, the flow rates from individual outlets were added for calculating the total flow rate of the emitter. In order to find the emitter exponent in the exponential pressure-flow rate relationship, the outlet flow rate was measured at eight pressure points by varying the inlet pressure between 20-200 KPa. The values of emitter exponent 'm' in the pressure-flow rate relationship for different variants were determined and compared with the recommended values by [5].

### B. *Anti-physical-clogging Performance*

Using the Discrete Particle Modelling in Ansys Fluent, a simulation was done to check the size and quantity of particles that escaping through the emitter outlets. This analysis was carried out on the two emitter flow path variants having one outlet and two outlets respectively. Particles of 'anthracite' material that are 'inert' were selected. Eight streams of particle input with randomized starting point were given as input normal to the inlet face.

The inlet velocity of the particles is assumed to be equal to that of the fluid inflow. The simulations were run for the two flow path variants, by taking the combination of five different particle sizes (0.0001, 0.001, 0.005, 0.01 and 0.1 mm) and three particle flow rates (1e-10, 1e-6 and 1e-5 kg/s). For each of the simulations, a particle track graph was generated by taking the 'particle residence time' as the parameter to colour the tracks. Finally, the 'average residence time' and the 'escape flow rate' of the particles were recorded.

## IV. Results and Discussion

The hydraulic and anti-physical-clogging performance assessment results of the novel cylindrical emitter are presented below.

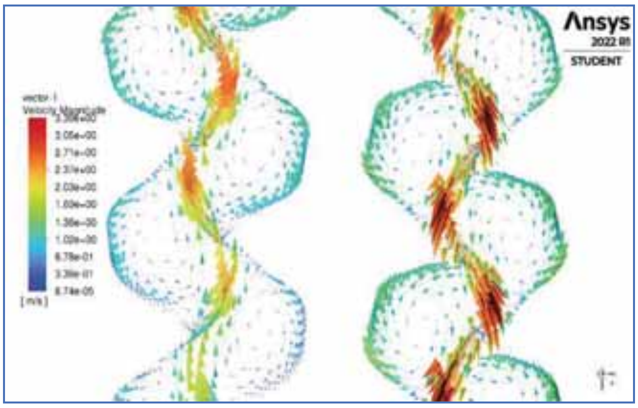
### A. Hydraulic Performance

The volume flow rate of 4 Lh-1 through emitter from one or two outlets is between 3.95 to 4.00 Lh-1 for an inlet pressure of 10.2 m (100 KPa), which is very close to the design flow rate (Table 1). While the velocity at the outlet increased compared to the inlet velocity in case of one-outlet variant, the same got reduced substantially to 0.311-0.324 ms-1 in the case of two-outlet variant.

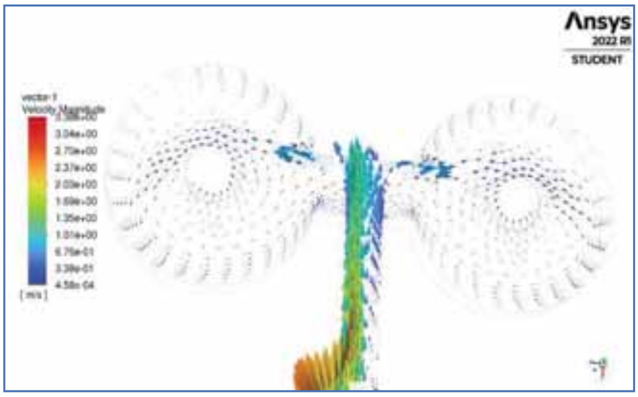
TABLE I.
Velocity and Flow Rate of Emitter Variants

| Flow Path | Inlet Velocity (ms$^{-1}$) | Outlet1 Velocity (ms$^{-1}$) | Outlet2 Velocity (ms$^{-1}$) | Outlet1 Flow Rate (Lh$^{-1}$) | Outlet2 Flow Rate (Lh$^{-1}$) | Total Flow Rate (Lh$^{-1}$) |
|---|---|---|---|---|---|---|
| One-outlet variant | 0.522 | 0.677 | -- | 3.988 | -- | 3.988 |
| Two-outlet variant | 0.527 | 0.324 | 0.311 | 2.080 | 1.913 | 3.992 |

The velocity vector graphs show maximum velocity in the zig-zag flow path at the tip of the dents combined with clear turbulent vortex formation on both the side walls, facilitating the flushing of any physical particles settled in the labyrinth zig-zag path (Fig.2a). There is helical flow formation at each outlet due to the circular shape of the outlet and the presence of cylindrical-shaped outlet projection (Fig.2b). This helical motion of flowing water is expected to dislodge and excrete the tiny physical particles that tend to settle at the outlets.



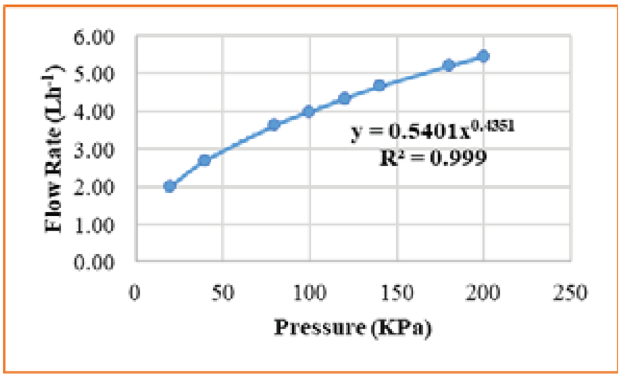

**(a)**



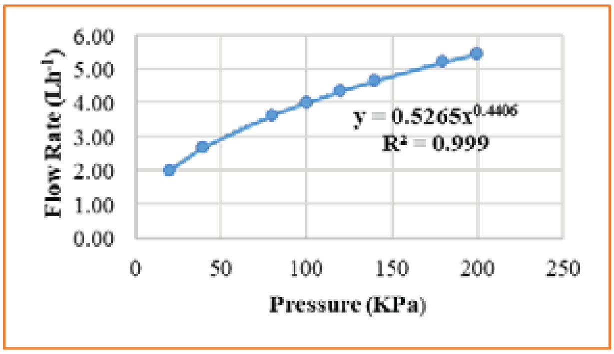

**(b)**

Figure 2. Velocity Field, (a) in the Labyrinth Path, and (b) at the Outlets

The pressure (p)-flow rate (q) relationship plotted for the 4 Lh-1 emitter paths indicated that the emitter exponent 'm' is well below the 0.5 limit recommended by the [5]. The regression coefficient values are > 0.95 indicating a good fit of the simulation data to the exponential regression model (Fig.3).




**(a)**




**(b)**

Figure 3. p-q relationship of, (a) one-outlet variant, and (b) two-outlet variant

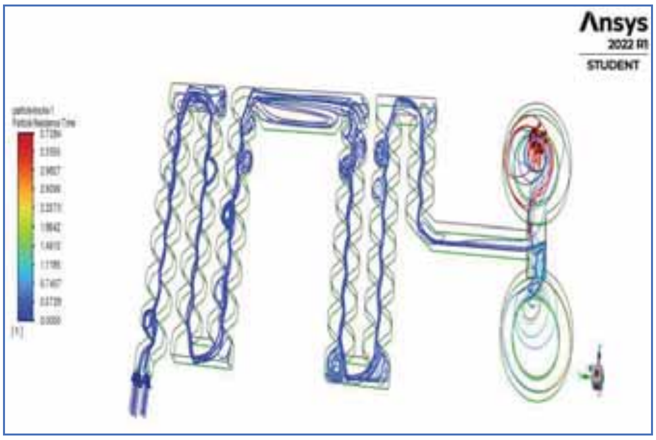
## *B. Anti-physical-clogging Performance*

Table 2 and Table 3 present the DPM results for one-outlet and two-outlet emitter variants, respectively. The residence time of the particles in the flow paths increased with increasing size of the particles, from 0.1 micron (0.0001 mm) to 100 micron (0.1 mm). However, varying the particle mass flow rate at the inlet (from 1e-9, 1e-6 and 1e-5 kg/s) did not affect any change in the particle residence time. For all the particle sizes and flow rate combinations, the particle escape rate is found to be 100%. The average particle residence time was 0.3779-1.036 s for the one-outlet variant and 0.2044-1.005 s for the two-outlet variant.

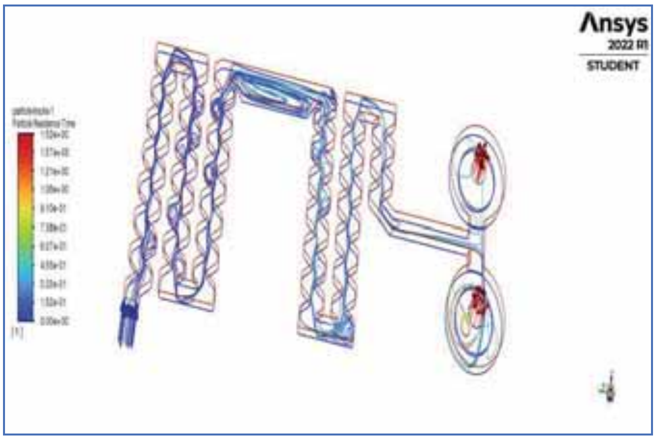
TABLE II.
AVG. RESIDENCE TIME AND % ESCAPE RATE OF PARTICLES FOR ONE-OUTLET VARIANT

| **Particle Flow Rate ($kgs^{-1}$)** | **Parameter** | **Particle Size** | | | | |
|---|---|---|---|---|---|---|
| | | **0.1 µm** | **1** | **5** | **10** | **100** |
| | | **0.0001 mm** | **0.001** | **0.005** | **0.01** | **0.1** |
| $1.00e^{-09}$ | Avg. residence time | 0.378 | 0.317 | 0.286 | 0.396 | 1.036 |
| | % Escape rate | 100 | 100 | 100 | 100 | 100 |
| $1.00e^{-06}$ | Avg. residence time | 0.378 | 0.317 | 0.286 | 0.396 | 1.036 |
| | % Escape rate | 100 | 100 | 100 | 100 | 100 |
| $1.00e^{-05}$ | Avg. residence time | 0.378 | 0.317 | 0.286 | 0.396 | 1.036 |
| | % Escape rate | 100 | 100 | 100 | 100 | 100 |

Large number of inlet vents, compared to the conventional cylindrical emitter designs, facilitate entry of water to the emitters even if some vents get clogged over time. Further, the helical motion created at the outlets 'lifts' the particles to be flushed out through the outlet holes. Thus, it is found that the proposed emitter designs are 100% efficient in excretion of physical particles in the size range of 0.1 to 100 microns. The particle tracks in the flow path indicated the residence time and the pattern of their movement along with the flowing water. Fig.4a presents the particle's tracks coloured by their residence time for the one-outlet variant. Similarly, Fig.4b presents the particle tracks through the flow path of the two-outlet variant. Both Fig.4a and Fig.4b are generated for the particle mass flow rate of 1.00e-05 kgs-1 and the particle size of 100 µm. These particle tracks clearly indicated that their movement is helical at the outlets. Some of the particles moved away from the high-velocity mainstream flow, expending more time in circular motion near walls in the labyrinth path. However, this phenomenon was not observed in case of particle size less than 100 µm.




**(a)**




**(b)**

Fig.4: Particle tracks coloured by their residence time in the flow path, (a) one-outlet variant, and (b) two-outlet variant

TABLE III.
AVG. RESIDENCE TIME AND % ESCAPE RATE OF PARTICLES FOR TWO-OUTLET VARIANT

| **Particle Flow Rate ($kgs^{-1}$)** | **Parameter** | **Particle Size** | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|
| | | **0.1 µm** | | **1** | | **5** | | **10** | | **100** | |
| | | **0.0001 mm** | | **0.001** | | **0.005** | | **0.01** | | **0.1** | |
| | | **Outlet1** | **Outlet2** | **Outlet1** | **Outlet2** | **Outlet1** | **Outlet2** | **Outlet1** | **Outlet2** | **Outlet1** | **Outlet2** |
| $1.00e^{-09}$ | Avg. residence time (s) | 0.204 | 0.350 | 0.286 | 0.580 | 0.29 | 0.408 | 0.272 | 0.47 | 0.472 | 1.005 |
| | % Escape rate | 16.67 | 83.33 | 50.00 | 50.00 | 50.00 | 50.00 | 33.33 | 66.67 | 16.67 | 83.33 |
| $1.00e^{-06}$ | Avg. residence time (s) | 0.204 | 0.350 | 0.286 | 0.580 | 0.29 | 0.408 | 0.272 | 0.47 | 0.472 | 1.005 |
| | % Escape rate | 16.67 | 83.33 | 50.00 | 50.00 | 50.00 | 50.00 | 33.33 | 66.67 | 16.67 | 83.33 |
| $1.00e^{-05}$ | Avg. residence time (s) | 0.204 | 0.350 | 0.286 | 0.580 | 0.29 | 0.408 | 0.272 | 0.47 | 0.472 | 1.005 |
| | % Escape rate | 16.67 | 83.33 | 50.00 | 50.00 | 50.00 | 50.00 | 33.33 | 66.67 | 16.67 | 83.33 |

## V. CONCLUSIONS

The novel emitter, with a design flow rate of 4 $Lh^{-}1$ and helical flow at the outlets, proved to be efficient in hydraulic and anti-physical-clogging performance. The simulation flow rates of 3.988 and 3.992 $Lh^{-1}$ are very close to the design flow rate. The emitter exponent values of the one-outlet and two-outlet variants are 0.4351 and 0.4406 respectively, which are well below the [5] stipulated limit of 0.5. While both the one-outlet and two-outlet variants were found with 100% particle escape rate, the average particle residence time was insignificantly higher (0.3779-1.036 s) for the one-outlet variant, compared to the 0.2044-1.005 s in case of two-outlet variant. Therefore, both the variants of the novel emitter developed are found to perform satisfactorily in terms of hydraulic and anti-clogging parameters. Laboratory testing of physical emitter models for validation of these simulation results forms the future scope of work.

## REFERENCES

[1] Ali, A.A.M. (2013). Anti-clogging drip irrigation emitter design innovation. European International Journal of Science and Technology, 2(8), 154-160.

[2] Bhagel, Y.K., Kumar, J., and Patel, V.K. (2021) CFD analysis of the flow characteristics of in-line drip emitter with different labyrinth channels, Journal of Institution of Engineers Ser. A, 102 (1), 111-119. https://doi.org/10.1007/s40030-020-00499-5

[3] BIS (2008) Indian Standard IS 13488 Irrigation Equipment – Emitting Pipe Systems – Specifications. Bureau of Indian Standards, New Delhi.

[4] BIS (2015). Indian Standard IS 14791 Prevention and treatment of blockage problem in drip Irrigation system – Code of Practice. Bureau of Indian Standards, New Delhi.

[5] BIS (2016). Indian Standard IS 13487 Irrigation Equipment – Emitters. Bureau of Indian Standards, New Delhi.

[6] Dazhuang, Y., Peiling, Y., Shumei, R., Li, Y. and Tingwu, X. (2007) Numerical study on flow property in dentate path of drip emitters, New Zealand Journal of Agricultural Research, 50(5), 705-712. https://doi.org/10.1080/00288230709510341

[7] Feng, J., Sun, H. and Li, Y. (2013) Visualizing particles movement characteristics in drip irrigation emitters with digital particle image velocimetry. Transactions of the CSAE, 29(13), 90-96. https://doi.org/10.3969/j.issn.1002-6819.2013.13.013

[8] Feng, J., Li, Y., Wang, W. and Xue, S. (2018) Effect of optimization forms of flow path on emitter hydraulic and anti-clogging performance in drip irrigation system. Irrigation Science, 36, 37–47. https://doi.org/10.1007/s00271-017-0561-9

[9] Li, Y., Feng, X., Liu, Y., Han, X., Liu, H., Sun, Y., Li, H. and Xie, Y. (2022) Research on hydraulic properties and energy dissipation mechanism of the novel water-retaining labyrinth channel emitters. Agronomy, 12, 1708. https://doi.org/10.3390/agronomy12071708

[10] Reethi, K., Mallikarjuna and Vijaya Raghu, B. (2015) Analysis of flow through a drip irrigation emitter. International Journal of Modern Engineering Research, 5(11), 57-64

[11] Weijing, Y., Zhengying, W., Huali, C. and Shengli, M. (2014) Optimal design and experiment for divided-flow emitter in drip irrigation. Transactions of the Chinese Society of Agricultural Engineering, 30(17), 117-124. https://doi.org/10.3969/j.issn.1002-6819.2014.17.016

[12] Wu, D., Li, Y., Liu, H., Yang, P., Sun, H. and Liu, Y. (2013) Simulation of the flow characteristics of a drip irrigation emitter with large eddy methods. Mathematical and Computer Modelling,58(3-4),497-506. https://doi.org/10.1016/j.mcm.2011.10.074

[13] Xu, T., Zhi, S., Yu, Q. and Zheng, E. (2022) Hydraulic performance and energy loss effect of pit structure optimized

drip irrigation emitter. Nature Environment and Pollution Technology, 21(2), 487-496.
[14] Yu, L., Li, N., Long, J., Liu, X. and Yang, Q. (2018) The mechanism of emitter clogging analyzed by CFD–DEM simulation and PTV experiment. Advances in Mechanical Engineering, 10(1), 1-10. https://doi.org/10.1177/1687814017743025
[15] Yu, L., Li, N., Liu, X., Yang, Q., Li, Z. and Long, J. (2019) Influence of dentation angle of labyrinth channel of drip emitters on hydraulic and anti-clogging performance. Irrigation and Drainage, 68, 256–267. " https://doi.org/10.1002/ird.2304
[16] Zhang, J., Zhao, W. and Lu, B. (2011) New Method of hydraulic performance evaluation on emitters with labyrinth channels. Journal of Irrigation and Drainage Engineering, 137, 811-815. http://dx.doi.org/10.1061/(ASCE)IR.1943-4774.0000365